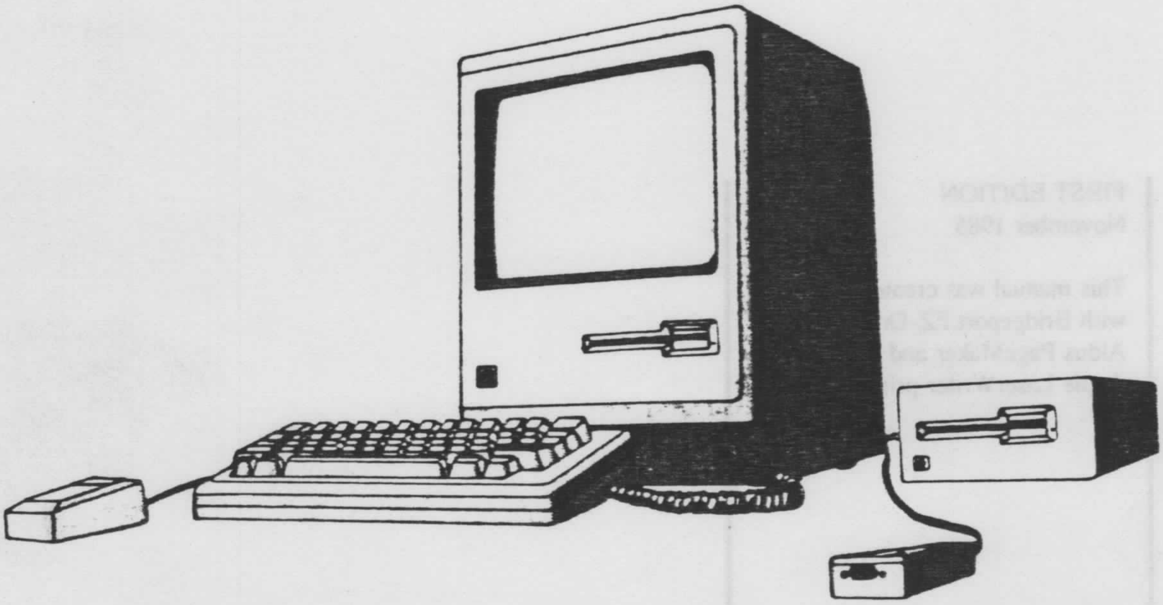
EZ-DRAFT™



For The Macintosh™

FIRST EDITION
November 1985

This manual was created with Bridgeport EZ-Draft, Aldus PageMaker and the Apple LaserWriter printer.

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About This Manual

This manual was created using figures generated by EZ-Draft. The figures used for examples are actual screen reproductions from the Macintosh and are intended to give you a good idea of what to expect while using EZ-Draft. Each figure gives you an example of the commands being carried out and the actual submenus and prompts that you will see while using EZ-Draft.

For an example of the plotting capability of the EZ-Draft system each chapter is preceded by an example plot that was created by EZ-Draft on either the Calcomp 1043 or Hewlett Packard plotters. For a complete listing of compatable plotters, hard drives and interfacing information please consult the Appendix: EZ-Draft Extras.

How To Use This Manual

Whether you are new to CAD or you have had experience with other systems, this manual will help you make the best use of your time and all of EZ-Draft's features.

The Manual is divided into six parts:

• *Before Starting* which covers how to use the manual, what you need and how to start up the EZ-Draft system.

• *Learning EZ-Draft* first teaches you the terminology used throughout this manual and the EZ-Draft system. Then beginning with the section titled "Entering Information" you will learn the basic skills for inputting information into EZ-Draft by using:
  - *Locate Menu* for placing point locations.
  - *Select Menu* for selecting geometry for modification.
  - *Global Commands* for moving around in EZ-Draft.

• *Setting Up Your Drafting Table* gives the procedures for setting up your work area and partitioning the drawing into individual views.

• *Drawing With EZ-Draft* Details those menus and commands that are used to create geometry.
  - *Lines Menu* with 34 different methods of creating lines.
  - *Arcs Menu* for creating for creating both circles and arcs.
  - *Other Menu* for creating closed lines and symbols.
  - *Detail Menu* for dimensioning and annotation.

• *Making Changes and Finishing Up* covers editing of a drawing and how to save it to a file disk.
  - *Edit Menu* provides powerful techniques for editing geometry as well as text.
  - *File Menu* covers all the commands necessary for storing, recalling and plotting your drawing.

• *EZ-Draft Extras* contains hardware specific information for plotting and hard disks, it also has a guide to short cuts in the EZ-Draft system.

Getting Started

To use EZ-Draft you will need:

- a 512K or 1 Meg Macintosh computer.
- the EZ-Draft software package
  - EZ-Draft Master disk
  - EZ-Draft System disk
  - EZ-Draft Data disk
- the User Identification (UID) Box, to be attached to the printer port of the Macintosh.
- a hard disk, see the appendix for the recommended disks.
- a plotter, which of the many plotters that EZ-Draft supports that you choose will depend on your requirements.

We assume that you are familiar with the Macintosh. If not, read *Macintosh* , the owners guide you recieved with the Macintosh. Before continuing any further with EZ-Draft.

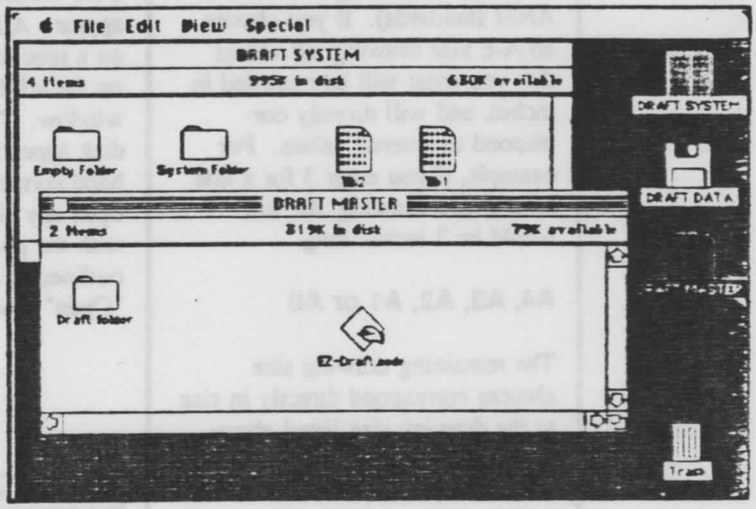
Starting Up

If you fill these requirements then you are ready to start up EZ-Draft.

- Write your EZ-Draft disks to your hard disk. You may use other arrangements such as Ram disk on 1 Meg or better Macintosh's.
- Make sure the User Identification (UID) box is attached to the printer port of the Macintosh.
- As needed open the window for the EZ-Draft Master disk (point on the icon and click the mouse, then choose "Open" from the desktop File Menu).
- Open the EZ-Draft code icon (point on the icon and click the mouse, then choose "Open" from the desktop File Menu).

In a minute or so you will see the opening screen, shown on the next page, and you will need to choose either a drawing size or "Get Drawing".

If you choose a drawing size a dotted line border, that represents your drawing, will appear. The dotted line signifies the active drawing view (geometry can only be created in an active view). It is recommended that you use this manual and the EZ-Draft Tutorial to learn the system before continuing on your own.



Drawing Size

A, B, C, D or E

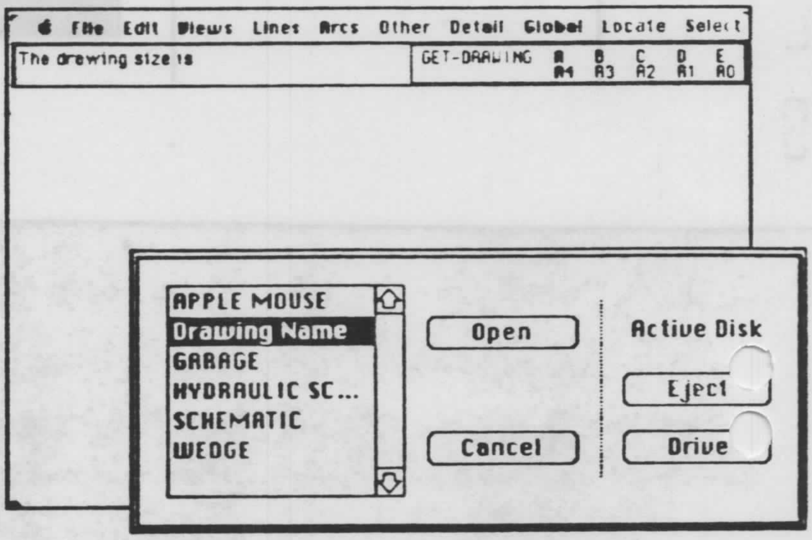
These choices are for standard size drawings (according to current ANSI standards). If you choose an A-E size drawing, all values that you enter will be recorded in inches, and will directly correspond to entered values. For example, if you enter 3 for a line on a B size drawing, the line would be 3 inches long.

A4, A3, A2, A1 or A0

The remaining drawing size choices correspond directly in size to the drawing size listed above them. However, when you select one of these sizes all of the entries will be read by the system as millimeters. For example, if you enter 3 for a line on a B size drawing, the line would be 3 millimeters long.

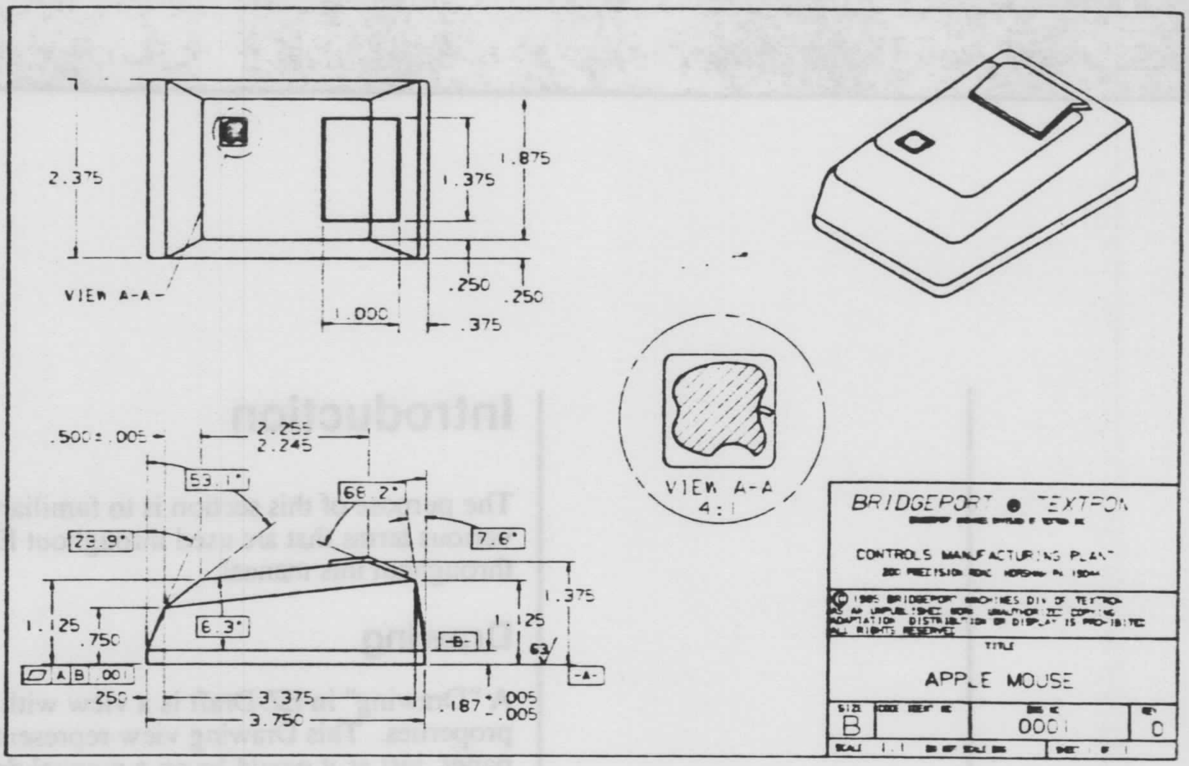
Get Drawing

This command allows you to open previously saved files. When you choose this command a the window shown below will appear. A list of files available on a specifc disk will be shown on the left hand side of the window. The name of the active disk appears in the upper right hand corner of the window. To open any of these files select it with the pointer, it will become outlined in black. Then choose "Open" from the list window and that file will be opened. You may select another disk to read files from by choosing "Drive" from the window. If your files are not on a disk currently in the sy[illegible]h choose "Eject" and insert the file disk.



The drawing size is

Chapter 1



Learning EZ-Draft

Introduction

The purpose of this section is to familiarize the user with the various terms that are used throughout EZ-Draft and also throughout this manual.

Drawing

A "Drawing" in EZ-Draft is a view with some special properties. This Drawing view represents an entire sheet of paper, just as it would be on a manual drafting board. However, with EZ-Draft your drawing will have much more versatility.

You begin by selecting a drawing size from the choices displayed on screen. The system will then display the Drawing border, which appears as a dashed line. The Orig (X=0, Y=0) is the lower left hand corner of the drawing. This Origin cannot be redefined. However, the border of the drawing view can be redefined as a different size sheet of paper at any given time. If the scale of the Drawing View is changed (default is 1:1), then each new view that you create will default to that scale.

The Drawing view always "draws" last, and never erases views already drawn. Therefore, lines that indicate connected views will remain displayed onscreen after the views have been drawn. All geometry, dimensioning, and annotation should be created within a view.

View

The "View" is the rectangular area of the drawing that is u defined. In most cases, a view created with EZ-Draft will correspond to the same view being designed or detailed or manual drafting board. For example, a Front View and a Top View of a part would be two distinct views in EZ-Draft. Another example would be a Section View of a part.

Each View has its own characteristics that can be changed or redefined independently of all other Views, giving the effect of a "drawing within a drawing". All elements extending beyond the View Border will be clipped (cut off) at the View Border. However, if you decide to enlarge the View Border, those clipped elements would again reappear.

The characteristics of a view fall into two categories, "View Control" and "View Definition".

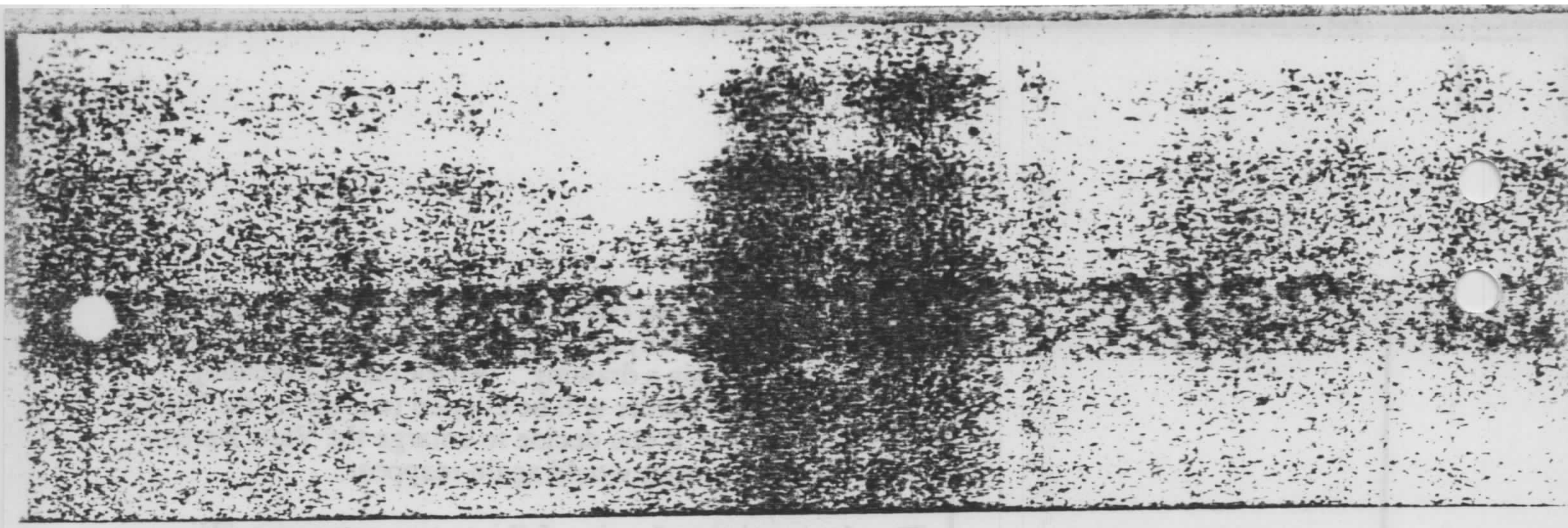
View Control

A view of a drawing can be used independently of all other views in the same drawing. The time required to select a particular element (point, line, arc, etc.) depends upon the number of elements in a particular view. Thus, using a new view to create geometry is faster than using a previously created view with many elements. In addition, it takes less time to draw or redraw a view with a few elements than one with many elements.

View Definition

Each item in this category is a value, location, or angle, that helps to define a View:

1. **View Border**
2. **View Origin**
3. **View Angle**
4. **View Scale**
5. **View Name**
6. **View Grid**
7. **View Trap Value**



Workview

Because each view of a drawing is independent, you can only work in one view at a time. The current view onscreen for creating and editing is the Workview. You may select a different view to work with at any time.

View Border

The View Border is the rectangular boundary of the view that appears as a dashed line onscreen. This boundary is entered when you first define the view, and can be changed at any time. All screen selections must occur inside the View Border. If you make a selection outside the View Border, it is ignored. With coordinate entry and editing commands, it is possible to create or move geometry and annotation outside of the View Border. If you do this, the geometry will still exis but it will not be displayed or plotted. If the View border is redefined to a larger size, this geometry will again be visible.

View Origin

The "View Origin is the location within the current view where the absolute coordinates are zero (X=0,Y=0). All point locations in the work view that you describe with absolute coordinates are determined from this "View Origin".

View Angle

The View Angle is the viewing angle of a drawing, as if it were three dimensional. This angle is entered for two reasons. Firstly, it is the same as the View Name. Secondly, when you use the "Projection" command in the "Other" me the type of projection used for each view depends upon this View Angle.

View Scale

The View Scale is the scale factor that is applied when the geometry is displayed. It is exactly like the scale factor on a manual drawing board (e.g.2:1). The geometry created onscreen is the correct (nominal) size; it is merely displayed with the scale factor applied.

For example, if you create a line 5 inches long and the view has a .5:1 scale factor, the line will appear 2.5 inches long. Similarly, the same view with a 1:1 scale factor will be 5 inches long when measured or dimensioned. If the drawing has a scale factor other than 1:1, then when you define (create) the view, it will be the same as the drawing's scale factor. Otherwise, the scale factor will be 1:1.

View Name

The "View Name" describes the "View Angle". This name is displayed onscreen, however it may also be turned off. If the view angle is explicitly entered, then that particular view will not have a view name.

View Grid

The "View Grid" is the user defined grid for a particular view. This grid can be polar (circular) or rectangular, and may be entered at any location in the workview. If the "Show Grid-On-Off" is On, then the point locations that you enter (while in the digitize mode) will automatically snap to the nearest grid point. If the "Show Grid-On-Off" is Off, then the point locations that you enter will be determined by the "Trap Value" (see Trap Value below).

The display of the grid is totally independent of the "Show Grid-On-Off" and can be turned On or Off at any time. In other words, the grid can be displayed while point locations do not snap to the grid. Similarly, point locations can snap to the grid even when it is not displayed. If grid points are too close to display quickly, then the grid will not be displayed, even if it is in the On position.

View Trap Value

The View Trap Value is used to "round off" point locations that you enter (by digitizing) with the snap grid turned Off. This prevents approximate screen locations from being entered inaccurately. For example, the default setting for A through E size drawings is 0.001 inch. If the Trap Value was 0.000001 inch, then a point entered by digitizing near X=1 would be 1.023746, whereas a point with 0.001 Trap Value would be at 1.024.

Layers

Layers are like transparent sheets of paper on a manual drafting board. In EZ-Draft, there are 256 different layers identified by number (from 1 to 256). These layers can be used to organize geometry and annotation for each drawing. For example, all of your notes for a drawing could be put on one layer, while geometry and dimensions for the same drawing could be put on two other layers. In each view, it is possible to control which layer (or layers) is displayed at any given time. It is also possible to display a certain range or combination of layers at one time.

Once your notes are completed on one layer, that layer's visibility can be turned Off so that the view redraws faster and elements are selected faster. In addition, you may use the "Select" menu to discriminate, allowing you to choose which layer(s) your selections are made from.

Colors

Six Colors (Red, Blue, Green, Yellow, Cyan, Magenta) are available for the display of geometry and annotation although only white and black will appear on the screen. The colors will show when the drawing is plotted. The default on the EZ-Draft system is a white screen with black to draw. However, this can be reversed to a black screen with white to draw using the "Invert Color" option under the "File" menu (see Chapter 2).

Coordinates

Coordinates are used to mathematically describe different locations in a view. The EZ-Draft system uses Cartesian (rectangular) Coordinates. This means that point locations are described with horizontal (X) and vertical (Y) distances from the current workview origin. For example, a point at X=2, Y=3 is two inches from the origin in the horizontal direction and three inches from the origin in the vertical direction.

The positive direction is to the right for horizontal (X) values, and to the top for vertical (Y) values. For instance, a point at X=2, Y=3 is two inches to the right of the origin and three inches up from the origin.

Another method of describing a point lcoation with EZ-Draft is to use Incremental Coordinates. Incremental Coordinates allow you to move by horizontal (IX) and vertical (IY) distances from the last point entered (the reference point).

Polar Coordinates enable you to describe point locations. With Polar Coordinates, the point is described by an angle (PA) and a length (LNG) from the last entered point (the Reference point).

Any of the three coordinate systems mentioned above can be used to describe point locations. However, all point locations are stored in the system as rectangular (also referred to as Absolute) coordinates.

Angle Specification

In the EZ-Draft system, angles are described in degrees. There are 360 degrees in a full circle. Positive angles move in a counter-clockwise direction, while negative angles move clockwise. Fractions of angles are described as decimal degrees. For example, an angle of 30 degrees, 30 minutes is entered as 30.5 degrees. An angle of 0 degrees corresponds to a positive horizontal direction and an angle of 90 degrees corresponds to a positive vertical direction. Similarly, an angle of 45 degrees would be in both positive horizontal and vertical directions.

Reference Point

The Reference Point point is the point location used for Incremental and Polar coordinate point descriptions. The Reference point is also used by the Angle Cross and is the point where the two lines of the Angle Cross intersect (see "Set ref point" under the Global menu).

Angle Cross

The Angle Cross is used just like the drafting machine arm on a manual drafting board. The "active" angle is the denser line of the two that make up the Angle Cross. The Angle Cross is further described in the "Locate" menu .

Points

A "Point" is a marker at an entered point location. It is displayed as a + onscreen, and marks individual locations for future reference.

Lines

A "Line" is a straight line between two point locations. There are many ways to describe and modify lines in the EZ-Draft system.

Infinite Lines

An "Infinite Line" is a special type of line. It is not actually infinite, but extends from its intersection with one view border to the next view border. Infinite Lines are often used as construction lines in the design process.

Arcs, Fillets, Circles

An "Arc" is a circular arc with a center at a specific point location, a radius, a starting angle, and an incremental angle.

A "Fillet" is a circular arc created to be tangent to two existi elements. Once created, a fillet is like any other arc.

A "Circle" is a circular arc whose starting angle is 0 degrees and whose incremental angle is 360 degrees. Thus, the starting and ending points of a circle are the same and are located at the angle position of 0 degrees.

Ellipses

An "Ellipse" is a conical seaction described by its major and minor radii, its center point location, its angle of rotation, its starting angle, and its incremental angle. This method of describing ellipses also makes it possible to describe partial and rotated ellipses.

B-Spline Curves

A "B-Spline Curve" is a smooth, fitted curve through a specific number of point locations. There must be at least 4 and no more than 50 point locations entered to describe a B-Spline Curve. The B-Spline is used like a "French Curve" on a manual drawing board, which fits a curve through specified points. A B-Spline Curve can also be used to draw unusual contours. The EZ-Draft system also has a powerful B-Spline editing capability that allows you to manipulate B-Spline points until the desired contour effect is reached.

Symbols and Symbol Instances

A "Symbol" is a collection of points, lines, arcs, and text. A "Symbol Instance" is an "occurrence" or an instance of placing a symbol in a view. A "Symbol Instance" can be a scaled, mirrored, and/or rotated version of a specified symbol.

"Symbol Instances" can only be selected by their origins. (The origin of a symbol is defined when you create the symbol. Similarly, the origin of a Symbol Instance is where the origin of the symbol is placed when the Instance is created). Symbol Instances can also be "smashed" into individual points, lines, arcs, and text.

A symbol with a number of Symbol Instances takes substantially less data space and makes element selection much faster than if the elements are created with "Move" or "Rotate" commands from the "File" menu.

Text

"Text" is comprised of a line of characters usually used as a note on a drawing. In EZ-Draft, text can be horizontal or angled, and can be right, left, or center justified. In addition, text lettering can be either straight or italic (slanted), and can appear in a variety of widths and heights.

Dimensions

A Dimension is a complete dimension including witness lines, arrowheads, dimension lines, and text (the text includes the dimension value and tolerances, as well as added text).

Other options can be used to create and edit dimensions, including a "smash" option that breaks down a dimension into individual lines and text.

Crosshatch

EZ-Draft provides 7 cross hatching patterns (see Appendix) and the facilities to design your own crosshatch patterns. "Crosshatching" is actually a symbol and a symbol instance. Using the "Smash" command, the Crosshatching can be turned into lines that can be edited.

Introduction

In the following chapters you will learn how to create, dimension, edit, modify and save both geometry and text. Before you can do that you will need to know how to get around in EZ-Draft.

This section covers those commands that are used interactively with the other EZ-Draft commands. When drawing with EZ-Draft you will use these commands to enter information, select and orient Geometry and to set specific parameters. These commands also enable you to modify how you view the drawing.

Number Entry

Numbers can be entered by using the keyboard or a numeric keypad. To complete each numeric entry, choose "Enter". To "Enter" use the keyboard, the numeric keypad, "Enter" under the "Global" menu or by double "clicking" the mouse. If you enter the digit incorrectly, press the "Backspace" key to delete the last digit, then re-enter the correct value. To cancel an entry, choose "Return" or another command, and the command in progress will not be executed.

Entering A Point Location

While using EZ-Draft you will often be prompted for points that describe or bound the Geometry that you are creating or modifying. Coordinates from the keyboard or commands in the locate menu allow you to describe points in the drawing view using a variety of different commands. Three general methods are Explicit Point Description, Length and Angle-Cross Control and Implicit Point Description.

Explicit Point Description

If a Point is described by Explicit entry, then the coordinates of that Point are entered by you, the user. These coordinates can be entered as Absolute, Incremental or Relative Polar Coordinates.

Absolute Coordinates X, Y

When a view is defined with Absolute Coordinates, the origin of the view is user defined. The origin is where absolute X=0, Y=0. Using the "X" and "Y" keys on the keyboard or the noted keys on the numeric keypad, enter a point with Absolute Coordinates X,Y by:

1. Entering the horizontal (X) distance from the origin.

2. Entering the vertical (Y) distance from the origin.

3. Choosing "Enter" to complete the command.

If the number is less than 0, then the horizontal (X) distance will be to the left of the origin, and the vertical (Y) distance will be down from the origin. Absolute Coordinates are like Incremental Coordinates except that the reference point is fixed at the view's origin.

If the Angle-Cross is On, then the actual point is determined by taking the perpendicular drop of the entered point to the active Angle on the Cross.

Incremental Coordinates IX, IY

When a point is described with Incremental Coordinates, the point is determined by taking the Reference point and adding the positive or negative incremental distances. Holding the option key down use the "X" and "Y" keys on the keyboard or the noted keys on the numeric keypad, enter a point with Incremental coordinates IX, IY by:

1. Entering the horizontal (IX) distance from the reference point.

2. Entering the vertical (IY) distance from the reference point.

3. Choosing "Enter" to complete the command.

If the Reference point is set before you choose "Enter", the new Reference point is used to determine the new point.

If the Angle-Cross is On, then the actual point is determined by taking the perpendicular drop to the active axis of the cross.

Polar Coordinates

Another method of describing points is with Polar Coordinates (angle and length) relative to the reference point. To determine the new point, start at the reference point and move the given length (L) along the given polar angle (A).

1. Enter the length (LNG) using the "L" key.

2. Enter the angle (PA) using the "A" key.

3. Choose "Enter" to complete the command.

If you set the reference point with the "Set Ref Point" command before choosing "Enter", then the new reference point is used to determine the new point.

If the Angle-Cross is On, then the actual point is determined by taking the perpendicular drop to the active axis of the cross.

Length And Angle Cross Control

If the Angle Cross is On (by choosing "Angle On Off"), then you can enter a length (L). This length can be moved along the active line of the Angle Cross. If the value is negative, then the direction is down or to the left. If the value is positive, then the direction is up or to the right. The reference point is at the intersection of the two Angle Cross lines.

When the Angle-Cross is On, all point locations are determined by the perpendicular drop to the active axis. The "Hor" selection determines the next point location by taking the vertical drop to the active axis. The "Ver" selection determines the next point location by taking the horizontal drop to the active axis.

Implicit Point Description

Implicit point description is found under the "Locate" menu. If a Point is described by Implicit entry, then the point location is determined by a screen position (in the current view) and the screen position mode.

The mode can be changed by choosing it from the menu. The active mode will be noted with a check mark. Discrimination is available for any command that requires specific geometry selection (See Discrimination in this chapter).

The modes of Implicit entry are:

1. **Digitize**, pointer or screen position.

2. **Endpoint**, endpoint of any selected element.

3. **Center**, of any selected line or arc.

4. **Point On**, point on a selected line or arc.

5. **Intersect 1**, intersection of the closest line and/or arc with the selected line and/or arc.

6. **Intersect 2**, intersection of two selected lines and/or arc.

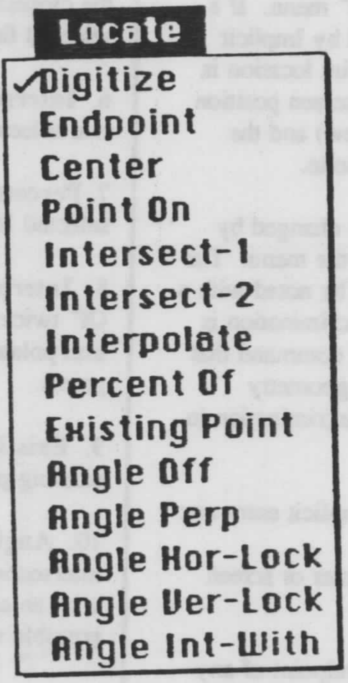
7. **Percent Of**, percentage of a selected line or arc.

8. **Interpolate**, choose "Percent Of" twice and EZ-Draft will interpolate between two selected points.

9. **Existing Point**, select an existing point.

10. **Angle-Int-With**, intersection of the Angle Cross with an existing line or arc (only possible withe Angle Cross On).

The first 9 commands in the "Locate" menu are used to describe point location for creating new geometry, modifing existing geometry or to set an origin for a Symbol Instance. The remaining commands are used for Angle Cross control which when in use is also used for point location.



Digitize

If the screen position mode is "Digitize", then the digitized position (that you choose) is used as the point location. The actual position is determined by the current screen position and the current trap settings (see Trap Value in Chapter). If the "Snap Grid On-Off" (see "Snap Grid On-Off" in Chapter) is On, then the point location is the nearest Grid point.

Endpoint

If the screen position mode is "Endpoint", then the endpoint of the nearest geometric element (e.g. point, line, arc, b-spline curve) will be used as the point location. First the system finds the nearest element. Then it determines which of the nearest element's endpoints is closest. This closest endpoint automatically becomes the point location. Any discrimination that has been set will be used by the system while finding the nearest element.

Center

If the screen position mode is "Center", then the center of the nearest line or arc will be used as the point location. For a line, the center is the midpoint of the line. For an arc, the center is the circular center of the arc. The system automatically locates the nearest line or arc and determines its center, which becomes the new point location. Any discrimination that has been set will be used by the system while finding the nearest line or arc.

Point On

If the screen position mode is "Point On", then the nearest Point location On the nearest line or arc will be used as the point location. If the nearest element is a line, then the point location is determined by a perpendicular drop to that line. If the nearest element is an arc, then the point location is determined by the intersection of the arc with a line from the center to the digitized location. In either case, if the screen position digitize is off of the line or arc, then the nearest endpoint will be used for the point location. Any discrimination that has been set will be used by the system while finding the nearest line or arc.

Intersect-1

If the screen position mode is "Intersect-1", the intersection of the nearest line or arc and the second nearest line or arc are used as the point location. Any Discrimination that has been set will be used by the system while finding the nearest line or arc.

Intersect-2

If the screen position mode is "Intersect-2", each of the two lines or arcs can be selected individually. Any discrimination that has been set will be used by the system while finding the nearest line or arc.

Interpolate

This command provides another way to describe a point location or a point representation (+) in the "Other" menu, and is an easy way of estimating values to find the center of a box without measuring or using construction techniques. To Interpolate:

1. After choosing "Interpolate" the prompt "Interpolate between" will appear.

2. Select two points (endpoints are usually used).

3. The system will prompt you to enter the desired "percent of" distance between the points you have selected. Most frequently, this is 50% for halfway between the points.

The system will automatically interpolate the "percentage" distances and points will appear at the requested distances. Notice that with the "Interpolate" command you can also enter negative percentage values or even a percentage greater than 100 percent.

Percent Of

If the screen position mode is "Percent Of" then the nearest line or arc is used with the percentage value (that you enter) to determine the point location. The nearest endpoint of the nearest line or arc is considered 0 %. The farthest endpoint is considered 100 %. Any descrimination is used by the system while finding the nearest line or arc.

Existing Point

If the screen position mode is "Existing Point" then the nearest existing point element will be used as the point location. Any Discrimination that has been set will be used by the system while finding the nearest point. Points in symbols can also be selected using this option.

Angle On or Angle Off

This command acts as a toggle switch to turn on or off the angle cross control (see Length and Angle-cross Control earlier in this section). By choosing the command "Angle On", the angle cross is activated, and the display will now read "Angle Off". When the command "Angle Off" is displayed and when chosen the cross control is turned off and the display will now read "Angle On".

Angle Perp

The "Angle Perp" command activates the angle that is perpendicular to the currently active angle. After you have chosen this command, no other information is necessary. This command functions regardless of whether the Angle-Cross is On or Off, but is only visible with the Angle-Cross On.

Angle Hor-Lock

This command can only be chosen when the angle cross is active. With the Angle-Cross On, "Angle Hor-Lock" locks the horizontal axis as the active axis.

Angle Ver-Lock

This command can only be chosen when the angle cross is active. With the Angle-Cross On, "Angle Ver-Lock" locks the vertical axis as the active axis when the Angle Cross is set at other than 0 degrees.

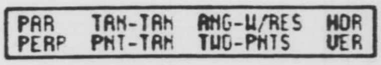
Angle Int-With

Angle Int-With is only active if the Angle-Cross is On. With the Angle-Cross On, Angle Int-With allows you to select a line or arc that intersects with the active line of the Angle-Cross. This point of intersection is the point location. If the element selected is an arc with only two possible intersection points, the intersection point nearest the screen position is used as the point location.

Any discrimination that has been set will be used by the system while finding the nearest line or arc.

Angle Entry Menu

This menu will appear anytime an angle value is needed. If you enter a number, then the other options disappear. If you describe the angle by numeric entry, then the number is in degrees (0-360), where 0 degrees is horizontal and 90 degrees is vertical.



```
PAR    TAN-TAN  ANG-W/RES  HOR
PERP   PNT-TAN  TWO-PNTS   VER
```

TAN-TAN

This command allows you to set the angle tangent to the two arcs or circles that you select. The side of the arc or circle that you digitize is where the tangent will be calculated. A prompt will also appear, asking if you want to select a new reference point or continue with the old reference point. If you want to continue with the old reference point choose "Enter".

PNT-TAN

This command allows you to set the angle through a point and tangent to an arc or circle that you select.

ANG-W/RES

This command allows you to set the angle of a new line with respect to the selected line's angle plus an increment.

TWO-PNT

This command allows you to set the angle as the angle between the positive horizontal axis and two points that you select.

PAR

This command allows you to set the angle parallel to an existing line that you select.

PERP

This command allows you to set the angle perpendicular to an existing line that you select.

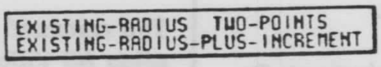
VER

This command allows you to set the angle to vertical (90 degrees).

HOR

This command allows you to set the angle to horizontal (0 degrees).

Radius Entry Menu



```
EXISTING-RADIUS   TWO-POINTS
EXISTING-RADIUS-PLUS-INCREMENT
```

This menu is used to enter radii. If you enter a numeric value, then the other options disappear.

EXISTING-RADIUS

This command allows you to use a radius equal to the radius of an existing arc or circle. Choose this command, then select the existing arc or circle with the cursor.

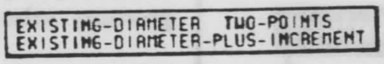
EXISTING-RADIUS-PLUS-INCREMENT

This command allows you to use an existing arc or circle's radius and enter an incremental value to be added to the existing radius. The new radius will have the value of the old radius plus the positive or negative incremental distance you enter.

TWO-POINTS

This command allows you to create a radius using the distance between two points that you enter. The two points can be entered using any of the point description methods. The distance between these two points is then used as the new radius .

Diameter Entry Menu



```
EXISTING-DIAMETER   TWO-POINTS
EXISTING-DIAMETER-PLUS-INCREMENT
```

This menu is used to enter diameters. If you enter a numeric value, then the other options disappear.

EXISTING-DIAMETER

This command allows you to use a diameter equal to the diameter of an existing arc or circle. Choose this command, then select the existing arc or circle which has the desired diameter.

EXISTING-DIAMETER-PLUS-INCREMENT

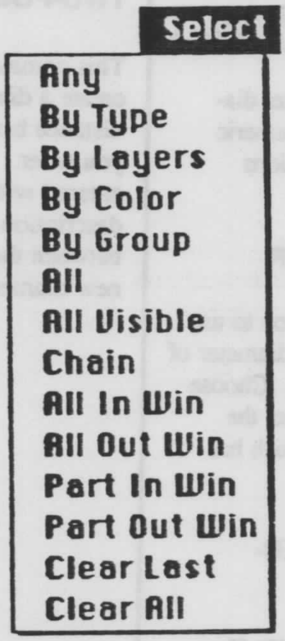
This command allows you to select an existing arc or circle's diameter and enter an incremental value to be added to the existing diameter . The new diameter will have the value of the old diameter plus the positive or negative incremental distance you enter.

TWO-POINTS

This command allows you to create a diameter using the distance between two points that you enter. The points can be entered with any of the point description methods. The distance between these two points is the new diameter .

Once your geometry is created you may want to manipulate it somehow, Edit, Change Color, Change Font, etc. . This menu allows you to select those elements of Geometry that you want to manipulate. Selected elements are highlighted to show that they have been selected.

This menu also allows you to Discriminate your selections. This means that you can choose a set of elements to make selections from and then only items in that set can be selected. There are four methods of Discrimination, **Layer, Type, Group** and **Color.**



Discrimination Commands

If **By Layer** discrimination is used, then only the layers that you specify are searched when geometry is being selected.

If **By Color** discrimination is used, then only the colors that you specify are searched when geometry is being selected.

Layer and Color discrimination remain On until you use the "Clear all" command to clear all discrimination.

If **By Group** discrimination is On, then only the Group-name that you specify is searched for geometry selection.

If **By Type** discrimination is On, then only the element types that you specify are searched for geometry selection.

If more than one method of discrimination is used, then all types of discrimination apply when making your selections. After each selection has been made in the discrimination menu, choose "Enter" to complete the command.

Multiple Selection Commands

These commands are used anytime multiple elements can be selected for modification. These commands are found under the Select menu after the Discrimination commands.

All

This command allows you to select All the elements in the current Workview. Choose All, then choose Enter. All of the elements will become Highlighted and are ready to be manipulated.

All Visible

This command allows you to select All Visible elements in the current Workview. If any portion of any element is visible then that entire element, even that portion not visible, will be selected.

Chain

A chain is a series of elements whose endpoints connect to form a continuous path. To select a chain of elements choose the "Chain" command and with the cursor, indicate the beginning of the chain of elements to be changed (these items will become highlighted).

All In Win

Choosing the "All In Win" command allows you to select all the elements completely enclosed inside a Window that you create with the mouse. This command will only allow you to select Points, Lines, and Arc/Circles. Any other type of element in that Window will remain unselected.

All Out Win

Choosing the "All Out Win" command allows you to select all elements Outside a Window that you create with the mouse. An element with any portion in the window will not be selected. This command will only allow you to select Points, Lines, and Arc/Circles. Any other type of element outside that Window will remain unselected.

Part In Win

Choosing the "Part In Win" command allows you to select elements that are either partially or completely within a window that you create with the mouse. This command will only allow you to select Points, Lines, and Arc/Circles. All elements completely outside that Window will remain unselected.

Part Out Win

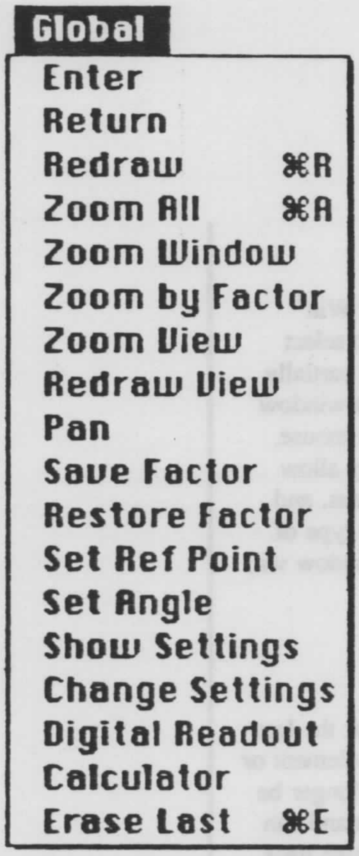
Choosing the "Part Out Win" command allows you to select elements that are either partially or completely Outside a window that you create with the mouse. This command will only allow you to select Points, Lines, and Arc/Circles. Any other type of element outside that Window will remain unselected.

Clear Last

This command will clear the last selection made and the element or set of elements will no longer be highlighted. This command can be repeated and it will step back thru the selections in the reverse order that they were made.

Clear All

This command will clear all selections and Discriminations. All the elements that had been selected will no longer be highlighted and all Discriminations will be turned off.



While working in EZ-Draft you may need to change how the drawing is displayed in order to place geometry with more accuracy. Or you might have to set a new font, color or reference to work with, this menu allows you to do this as well as "Enter" and "Return" without using the keyboard. "Global" commands can be executed while another command is in progress, information entered for the command in progress is saved and later restored after the "Global" command is completed. The prompt screen is also restored to help you quickly recover saved information.

In most instances, only one "Global" command can be executed at a time. Only Perp concurrent commands can be executed while another "Global" command is in progress.

Enter

Frequently while you are using EZ-Draft it is necessary to choose "Enter" to indicate that data entry has been completed. For instance, when you are entering a point by its coordinates, choosing "Enter" is required to indicate that the description is complete. Also, if a command allows more than one element to be selected, you must choose "Enter" to indicate that all desired selections have been ma The enter key on the keyboard a the numeric keypad serve the same purpose. "Enter" can also be done by double clicking quickly on the mouse.

Return

If "Return" is chosen in the middle of a command, the command is aborted and no additional changes are made in the drawing. The return key on the keyboard and the numeric keypad serve the same purpose.

Display Commands

The "Display" commands allow you to change the way you "see" the drawing onscreen. The information in the drawing does not change, but your perspective does, allowing you to see the information from closer up, further away, or centered at a different point. This is much like leaning toward, leaning away, or scanning across a drawing on a drafting board.

Redraw

This command allows you to redraw the image currently onscreen. The "Redraw" command can be used any time you suspect there might be information not visible in the drawing due to erasures, trimming, or editing. After choosing "Redraw", no other information is required. A redraw is occasionally necessary due to the technology of the monitor (Raster technology). For example, if two lines are on top of each other and one of them is erased, they will both apepar to be erased until the image is redrawn. This command can also be chosen by holding down the Command key and pressing the "R" key.

Zoom All

This command allows you to zoom the entire drawing to fill the screen. For example, if you have zoomed a portion of your drawing and wish to restore the entire drawing, simply choose "Zoom All". After choosing this command, no other information is required. This command can also be chosen by holding down the Command key and pressing the "A" key.

Zoom Window

The "Zoom Window" command allows you to zoom in on a portion of your drawing. The portion of the drawing that will fill the screen is identified by a rectangular box that you can indicate with a few simple steps:

1. Choose "Zoom Window".

2. Indicate the lower left hand corner of the box with the pointer (the lowest possible left hand corner of the box would be at the bottom left hand corner of the screen).

3. Next, move the pointer diagonally upward until the box is the desired size (you will be able to see the box moving).

4. Indicate the upper right hand corner of the box with the pointer.

The "close-up" view that you have indicated will automatically be redrawn. The system assures that all elements in the box you have indicated will be visible when the view is redrawn. If the height/width ratio of the desired area is greater than the height/width ratio of the screen area, you may want to indicate a box that will center the desired area onscreen.

Zoom by Factor

This command allows you to zoom in on the current image by a specified factor. The system will prompt you to enter the factor to zoom in by, or to choose "Enter" (which defaults to a zoom factor of 2). The current image will be enlarged by the zoom factor (number of times) you have indicated. The lower left hand corner point will remain fixed. "Return" aborts the command.

Zoom View

Choosing the "Zoom View" command allows you to zoom a selected view. The view you select with the cursor will then fill the screen.

If only one view is visible, (e.g. if you are already zoomed in on a portion of a view) simply indicate any part of the current view and it will fill the screen. If you zoom a view which is not the currently active view, then you can not create any new geometry onscreen, and existing geometry can not be selected since it is not the active workview. To correct this, select this view as the workview.

Redraw View

Choosing the "Redraw View" command allows you to redraw a single view selected with the pointer. The indicated view will automatically be redrawn. "Redraw View" saves time if there is a lot of geometry in other views that does not require redrawing.

Pan

This command allows you to pan across a drawing. Choose "Pan", then with the pointer, indicate where to pan from and then indicate where to pan to.

Save Factor

This command allows you to save the current display settings with a name, so that you can return to this zoom factor at a later time. After you choose "Save factor", the system will prompt you to enter a name to identify the current settings to be saved. An example of a name might be "Corner" if the corner of a part was zoomed at the time. After you enter the name, the settings are saved, and you will be able to return to this screen by using the "Restore Factor" command. This command is not yet implemented.

Restore Factor

This command allows you to retrieve display settings for zoom locations which were previously saved using the "Save factor" command. To execute the "Restore factor" command, simply enter the name used to save the settings. If the name is valid, then the system will automatically redraw the view using the saved settings. This command is not yet implemented.

Set Ref Point

The "Set Ref Point" command allows you to set the reference point, which is used as:

1. The base point for Incremental Coordinate point descriptions.

2. The base point for Polar Coordinate point description

3. The location of the Angle-Cross (see Angle-Cross in Chapter)

Using the "Locate" menu you can place the reference point anywhere on the screen by digitizing or by using existing geometry. If you terminate this command with a "Return", then the Reference point will automatically be set at the last entered point.

Set Angle

The "Set Angle" command allows you to set the angle for the Angle-Cross. First, you must enter the angle. If the Angle-Cross is On, it will automatically be turned Off. The Angle-Cross is not turned On with this command. The Angle is turned back On by choosing the "Angle On Off" command in the "Locate" menu.

Show Settings

This command allows you to display the current values of Angle, Reference point, active View, Layer, Color and Curve font. Simply choose the "Show Setting" command, and the values will automatically be displayed in the prompt area of the screen.

Change Settings

The "Change Settings" command allows you to set the "Layer", "Group", "Color" or "Curve-Font" when creating geometry.

After choosing a parameter from "Change Settings", the above submenu will appear, allowing you to change the parameter to the desired setting. When you have selected the new setting, choose "Enter" to complete the command. To exit the "Change Settings" submenu without changing any of the settings, choose "Return" or another command.

LAYER

This command allows you to select one of the 256 "transparent" layers as the active layer that will be used for subsequent geometry and detailing. Only one layer can be drawn on at a time, although any combination of layers can be visible at the same time (this is determined with the layer command in the "Visible" submenu). By using layering, you can temporarily blank out other layers, allowing you to concentrate on specific parts of very detailed drawings (e.g. subassemblies). Choose "Layer", then enter the number of the layer (the values of the layer are from 1 to 256) you wish to make active, then choose "Enter" to complete the command.

GROUP

This command allows you to assign a Group-name to all subsequent geometry and detailing that you create. To do this choose "Group", then enter the name to be used as the Group-name and then choose "Enter" to complete the command. This Group-name will be used until you select another Group-name, or clear the current Group-name by choosing "Group" followed by "Return" (without a new name).

COLOR

The Macintosh only displays in black and white, however EZ-Draft has the ability to keep track of six colors, this very useful for color plotting.. This command allows you to choose from the six available colors Red, Green, Yellow, Blue, Magenta and Cyan. Desired colors can be chosen by name from the "Color" submenu with the cursor. Choose "Color", then choose the desired color and then choose "Enter" to complete the command. The chosen color will now be used all subsequent geometry and detailing until a new color is chosen.

CURVE-FONT

This command allows you to set the Curve-Font that you will use in subsequent geometry creation. The font is identified by a number that you enter. Available Curve-Fonts are:

1. Solid
2. Dashed
3. Long Dash
4. Dash / Dot
5. Long Dash / Short Dash
6. Long Dash / Short Dash / Short Dash
7. Dot

Simply enter the desired Curve-Font by number, then "Enter" to complete the command.

Digital Readout

```
ABS-COORD  POLAR-COORD
INC-COORD  NONE
```

The "Digital Readout" command digitally displays a point location's coordinates, this slows the response of the EZ-Draft system, because the display is always being refreshed.

ABS-COORD

When you choose this command the readout will be given in terms of absolute coordinates (X, Y). The readout will start after selecting your first point location.

POLAR-COORD

When you choose this command the readout will be given in terms of polar coordinates (L, A). The readout will start after selecting your first point location.

INC-COORD

When you choose this command the readout will be given in terms of incremental coordinates (IX, IY). The readout will start after selecting your first point location.

NONE

Choosing "None" turns Off any of the digital display options.

Calculator

```
SAVE  SQR   ARC  TAN
USE   SQRT  COS  SIN
```

This menu gives you the ability to Calculate a number, save the number, and then use it as a number entry. The Calculator submenu includes standard arithmetic, square, square root, trigonometric, and inverse trigonometric functions. Remember all operations are performed in the order that you enter them.

Calculator Functions

The arithemtic functions in the "Calculator" submenu consist of: Add (+),Subtract (-), Multiply (*) and Divide (/). These selections all function like a standard pocket calculator. Your selections are made on the keyboard or using the numeric keypad.

SQR

Choosing "Sqr" calculates the square (the number multiplied by itself) of the number to be entered.

SQRT

Choosing "Sqrt" calculates the square root of the number to be entered.

COS

Choosing "Cos" applies the trigonometric function Cosine to the number to be entered in degrees.

SIN

Choosing "Sin" applies the trigonometric function Sine to the number to be entered in degrees.

TAN

Choosing "Tan" applies the trigonometric function Tangent to the number to be entered in degrees.

ARC

"Arc" puts the calculator into the inverse trigonometric mode. This means that the next trigonometric function selected (SIN, COS, or TAN) will actually be the inverse function (ARC SIN, ARC COS, or ARC TAN).

SAVE

Choosing "Save" saves the calculated number so that it can be used as a number entry. "Enter" or (=) must be chosen before saving a calculated number to ensure that all operations are completed.

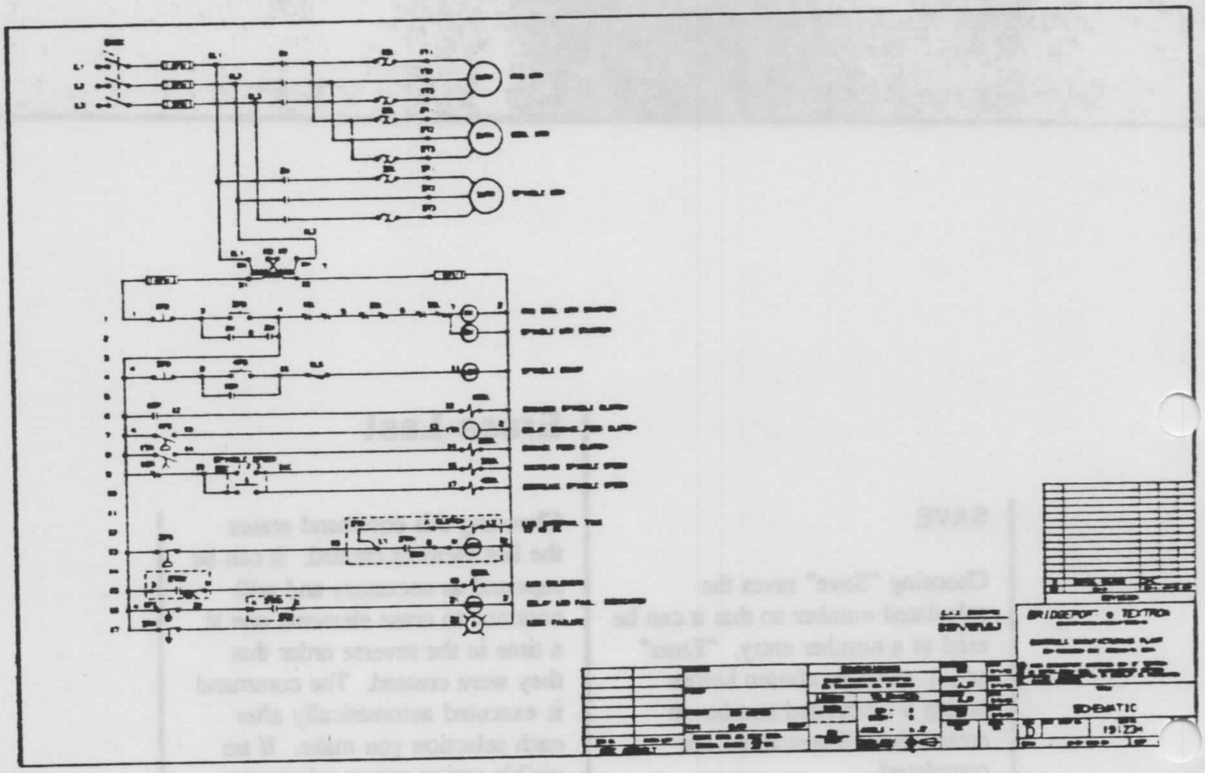
USE

By choosing "Use" you can retrieve a saved number in the calculator and use it as a number entry. In order for this command to be executed, you must be in a number entry process. The saved number can only be used one time.

Erase-Last

Choosing this command erases the last element created. It can be repeated as necessary and will continue to erase elements one at a time in the reverse order that they were created. The command is executed automatically after each selection you make. If no visible action occurs when you select this command, the element is either out of the view or has already been erased. This command can also be chosen by holding down the Command key and pressing the "E" key.

Chapter 3



The [illegible] command erases the [illegible] created. It can be [illegible] as necessary and will [illegible] erase elements one at a time in the reverse order that they were created. The command is executed automatically after each selection you make. If no visible action occurs when you select this command, the element is either out of the view or has already been erased. This command can also be chosen by holding down the Command key and pressing the "E" key.

SAVE

Choosing "Save" saves the calculated number so that it can be used as a number entry. "Clear" [illegible] completed.

USE

By choosing "Use" you can retrieve a saved number in the calculator and use it as a number entry. In order for this command to be executed, you must be in a number entry process. The saved number can only be used one time.

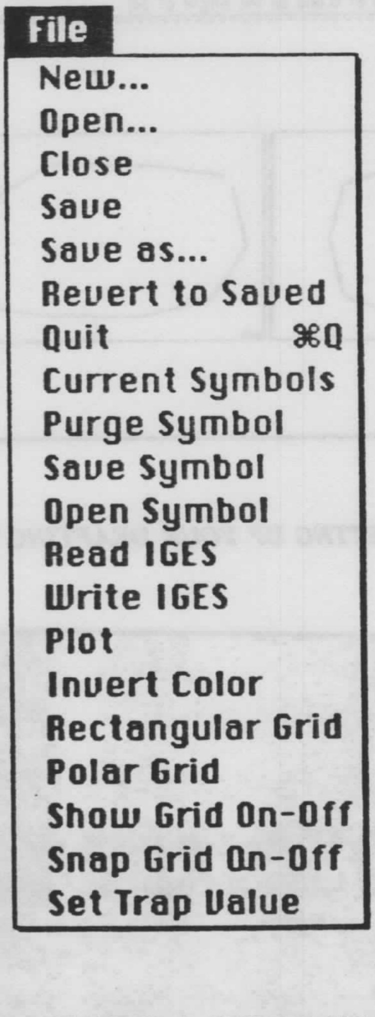
Setting Up Your Drafting Table

Setting Up Your Drafting Table

Introduction

Before starting to draw, you will need to set up your table top and drawing area. The commands in this chapter enable you to set the parameters for the whole drawing using commands from the "File" menu. The commands in the "Views" menu will help you divide your drawing into the individual views that you need to use.

File Menu



There are 21 choices under the "File " menu, the last 6 of which will be discussed in this chapter. These 6 commands allow you to set the parameters, such as screen color and trap value for the drawing area. The rest of the "File " menu will be discussed in Chapter 4 "Making Changes and Finishing Up".

Invert-Color

Choosing this command changes the color of the screen's background. The default for the Macintosh is black lines on a white background. However, if you wish to have white lines on a black background, you can by choosing "Invert-Color". This only changes how you see the screen and it has no affect on the "COLOR" (see p. 39 Change Settings).

To return the screen to black lines on a white background choose "Invert-Color" again and the screen will return to the original setting.

Rectangular Grid

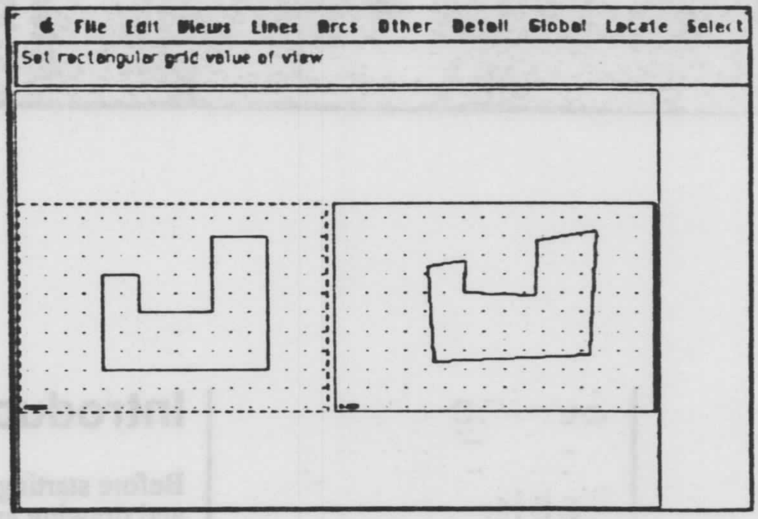
The "Rectangular Grid" command allows you to define a rectangular grid in a selected view. To define a rectangular grid:

1. Select the desired view.

2. Enter the X and Y spacing for the grid points.

3. Enter a starting (center) location. After the grid is drawn you are returned to the Menu Bar.

The "Show Grid On-Off" and "Snap Grid On-Off" are automatically turned On when you define a grid. If a previous grid was defined in the view, it will be replaced by the newly defined grid.


File Edit Views Lines Arcs Other Detail Global Locate Select

Set rectangular grid value of view

Polar Grid

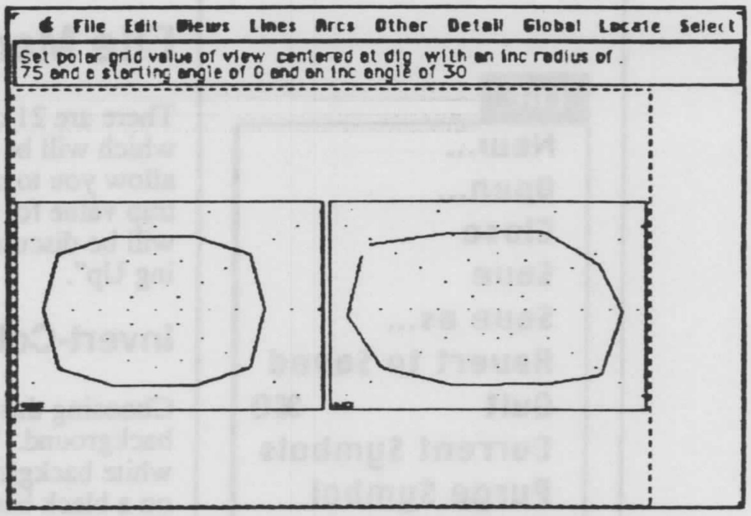
This "Polar Grid" command allows you to define a polar or circular grid in the selected view. To define a polar grid:

1. Select the desired view.

2. Enter a starting (center) location.

3. Enter the incremental radius of grid points.

4. Enter the starting angle of the grid.


File Edit Views Lines Arcs Other Detail Global Locate Select

Set polar grid value of view centered at dig with an inc radius of .
75 and a starting angle of 0 and an inc angle of 30

5. Enter the incremental angle between grid points.

6. Answer yes("Enter") or no("Return") at the prompt, "Do you want full screen polar grid display?". The Default is a single (first) radius display.

The "Show Grid On-Off" and "Snap Grid On-Off" are automatically turned On when you define a grid. If a previous grid was defined in the view, it will be replaced by the newly defined grid.

Show Grid On-Off

This command allows you turn On or Off the visibility of a selected view if a grid has been defined. After choosing this command, select the desired view and then "Enter" to complete the command. The Show Grid On-Off command does not turn on the Snap-To grid.

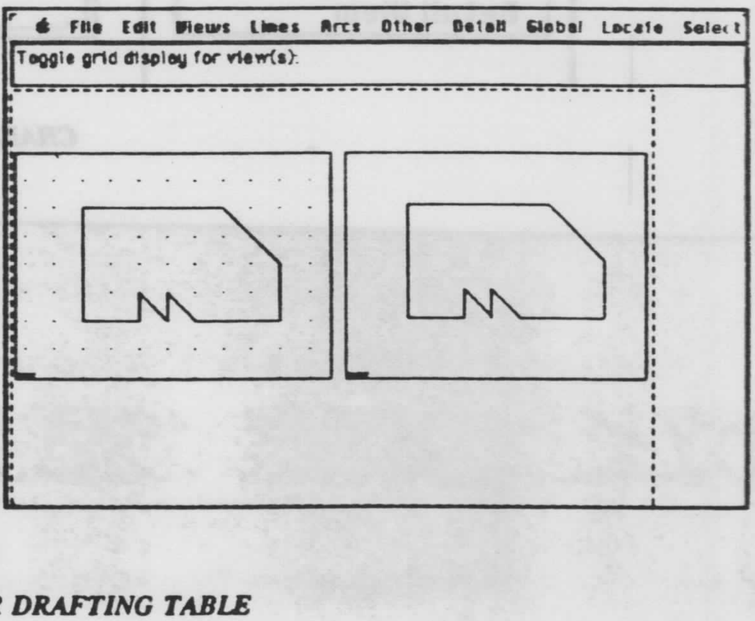
Snap Grid On-Off

The "Snap Grid On-Off" command allows you to make all future digitizes in the selected view(s) snap-to the currently defined grid. The grid does not have to be displayed in order for the snap-to feature to be On. This command also allows you to turn Off the snap-to feature of the grid for all future digitizes in the selected view(s). The grid can still be displayed for reference purposes, but future digitizes will not snap-to the grid. After choosing this command, select a view and then "Enter" to complete the command.

Set Trap Value

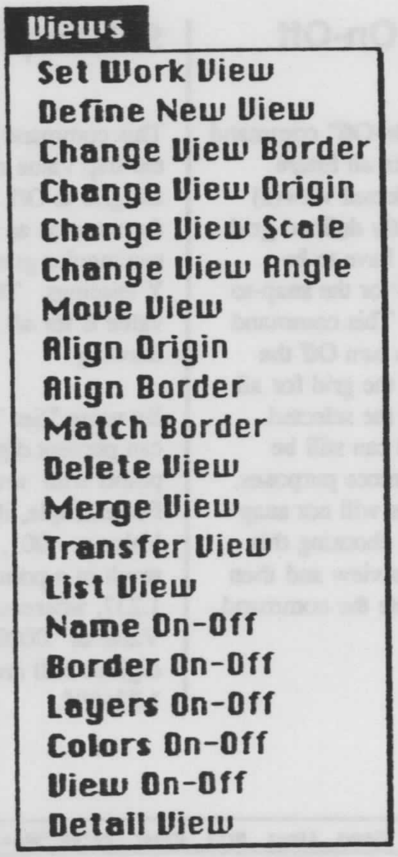
This command allows you to set the trap value of digitizes when the grid is Off. The Trap Value is the same as an invisible rectangular grid with equal X and Y spacings. The selected trap value is for all views of the drawing

By using "Set Trap Value", you can prevent digitizes from creating points with unrealistic accuracy. For example, if you set the Trap Value at .001, a Digitize can result in a point being defined at 1.237, whereas if you set the Trap Value at .000001, the same digitize will create a point at 1.236895.



Toggle grid display for view(s):

When you started the system up you defined the size of the drawing. In EZ-Draft you can divide the drawing into as many "Views" as are required to properly represent the workpiece. The "Views" menu allows you to define and modify these specific views on a drawing.



Set Workview

This menu allows you to select which view will be the current view for creating and editing elements.

Define New View

This command allows you to define specific views of a drawing.

To define a view:

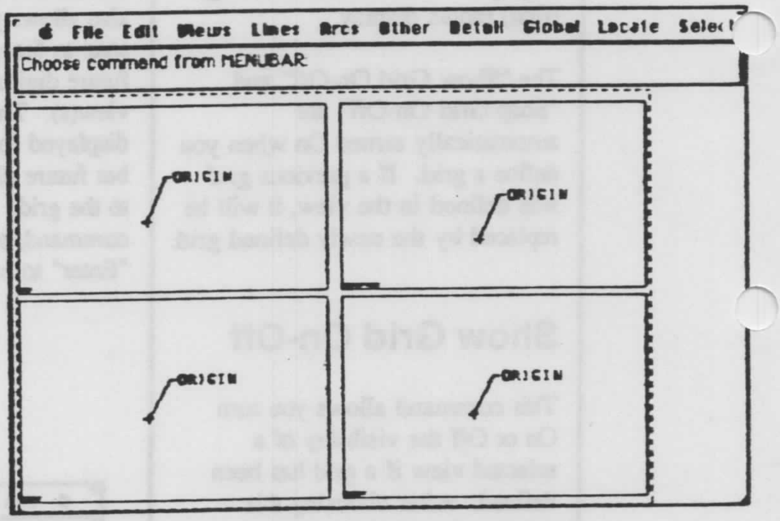
1. Indicate the two opposite corners of the view border.

2. Indicate the location of the view origin. (This is where absolute coordinates X=0 and Y=0 will be located).

3. Finally, select a view angle from the choices displayed (e.g. top, bottom, left, right etc.) This new view angle can be used later to generate geometry for projections. All other characteristics of the view are set to pre-defined conditions, but can be changed with various commands in the "Views" menu.

Change View Border

This command allows you to redefine the border of a view. The border is redefined by indicating the two opposite corners of the new view border.



Choose command from MENUBAR

Change View Origin

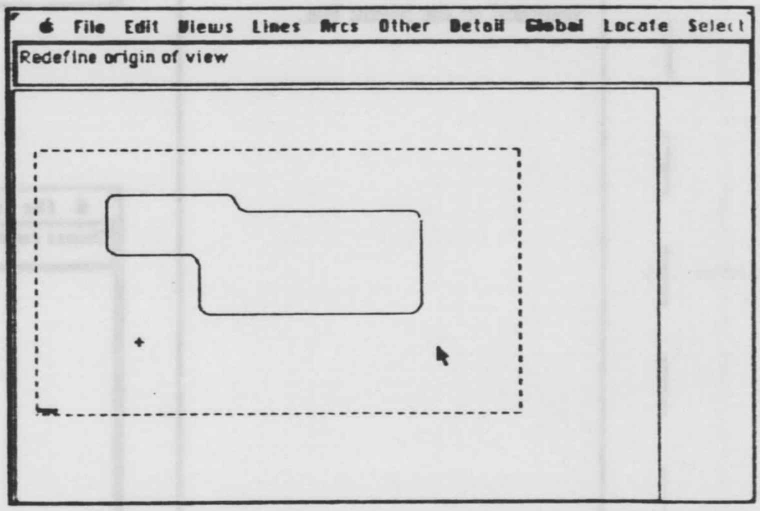
This command allows you to redefine the origin of a view. The origin is redefined by indicating the position where absolute coordinates X=0, Y=0 will be located.

Change View Scale

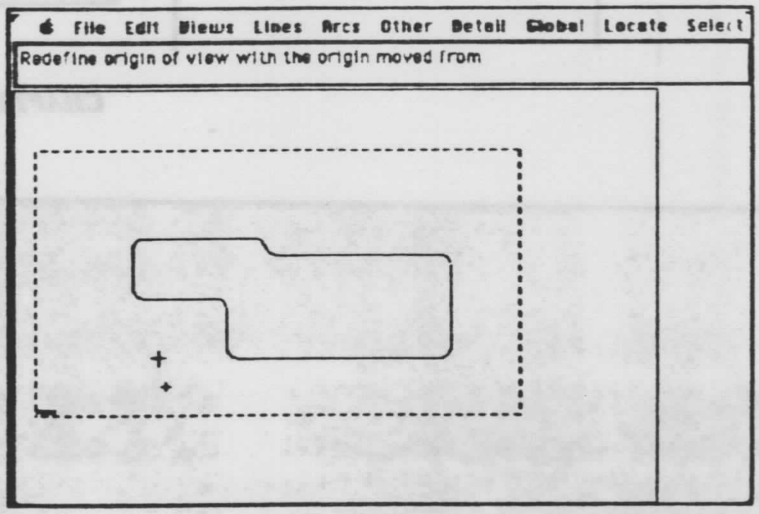
This command allows you to change the scale of the selected view. With this command you can also draw parts larger than the actual paper size, or enlarge small parts for better readability.

Change View Angle

This command allows you to redefine the view angle of a view. You can select the new view angle from the pre-defined views Top, Bottom, Front, Back, Left, Right, Isometric, or enter the new view Explicitly. The view angle choices are shown below. The view angle is important since it is used to create geometry from the projection command.



Redefine origin of view



Redefine origin of view with the origin moved from

Move View

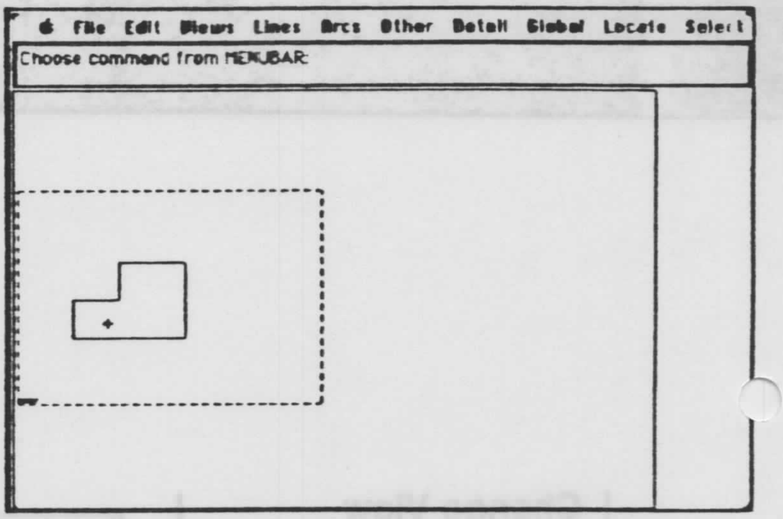
This command allows you to move a selected view from a selected reference point to a selected destination point.

1. Select the view to be moved.

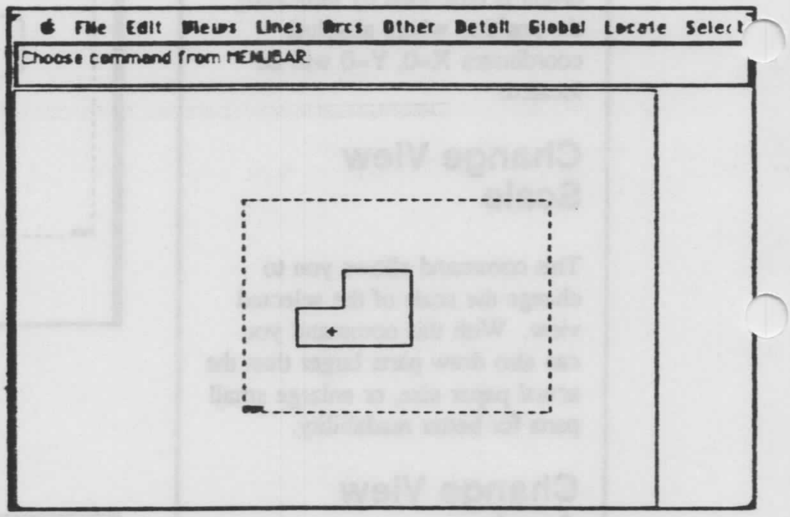
2. Enter the point to be moved from.

3. Enter the point to be moved to.

After the view has moved you are returned to the Menu Bar.


File Edit Views Lines Arcs Other Detail Global Locate Select

Choose command from MENUBAR:


File Edit Views Lines Arcs Other Detail Global Locate Select

Choose command from MENUBAR:

Align Origin

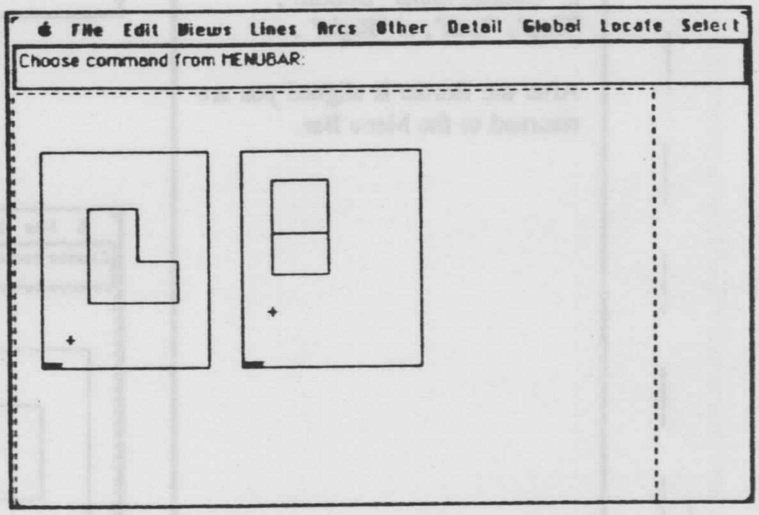
This command allows you to align the origin of a selected view with the origin of another selected view. The alignment is in either the horizontal or vertical direction.

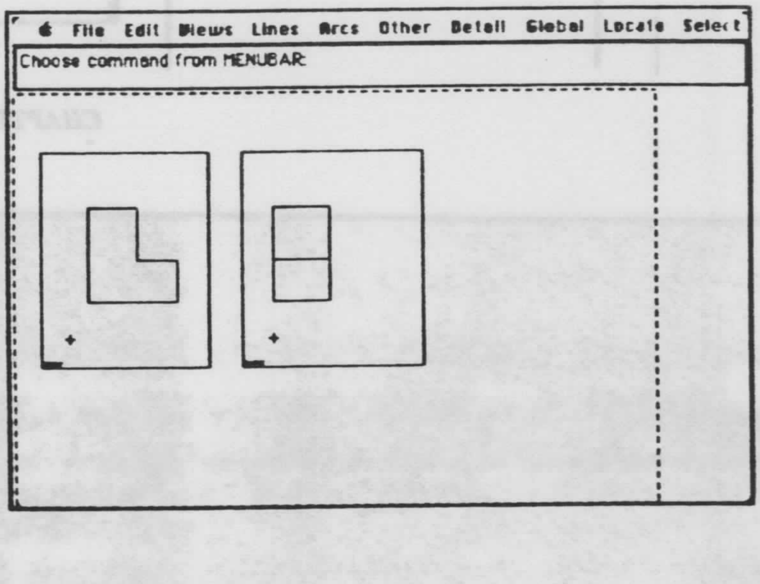
1. Select the view with the Origin to be aligned.

2. Select the view to align the origin with.

3. Choose either "Move-Vertical" or "Move-Horizontal".

After the Origin has moved you are returned to the Menu Bar.


File Edit Views Lines Arcs Other Detail Global Locate Select
Choose command from MENUBAR:


File Edit Views Lines Arcs Other Detail Global Locate Select
Choose command from MENUBAR:

Align Border

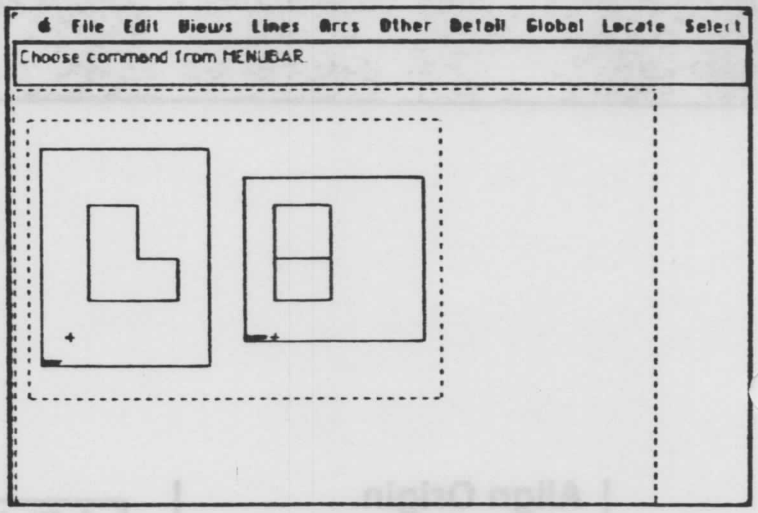
This command allows you to move one side of a selected view border and align it with the same side of another selected view border. The alignment choices are "Bottom", "Top", "Left", or "Right" .

1. Select the view to be aligned.

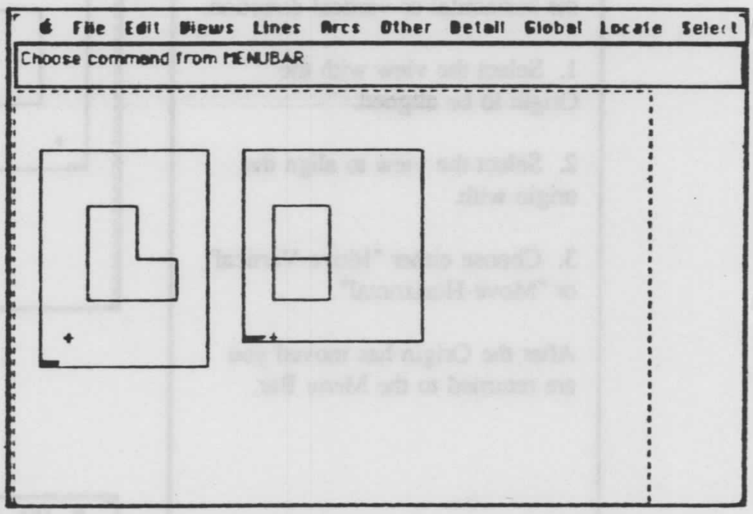
2. Select the view to align Border with.

3. Choose either "Bottom", "Top", "Left", or "Right" .

After the Border is aligned you are returned to the Menu Bar.


File Edit Views Lines Arcs Other Detail Global Locate Select

Choose command from MENUBAR.


File Edit Views Lines Arcs Other Detail Global Locate Select

Choose command from MENUBAR.

Match Border

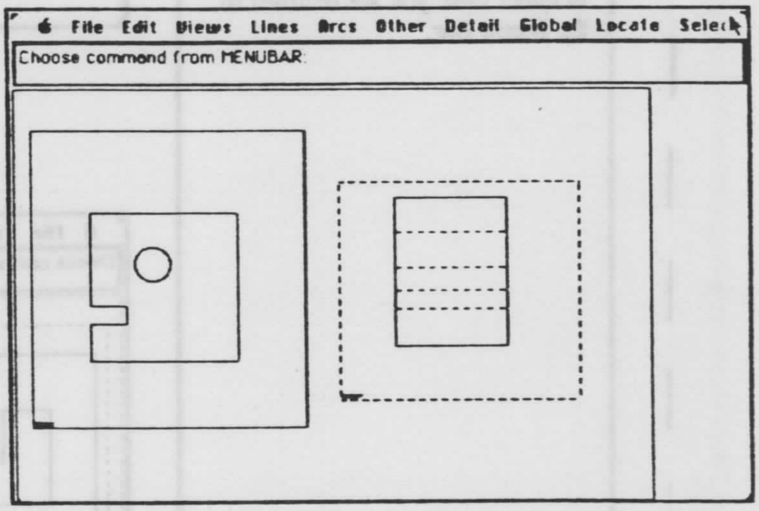
This command allows you to align either the top and bottom (horizontal) or left and right (vertical) sides of a selected view border with another selected view border.

1. Select the view to be aligned.

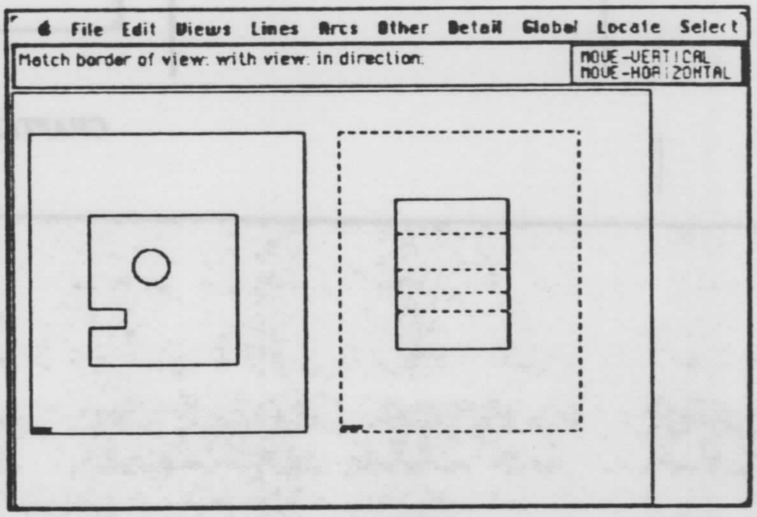
2. Select the view to be Matched.

3. Choose either "Move-Horizontal" or "Move-Vertical".

After the Border is aligned you are returned to the Menu Bar.


File Edit Views Lines Arcs Other Detail Global Locate Select

Choose command from MENUBAR:


File Edit Views Lines Arcs Other Detail Global Locate Select
MOVE-VERTICAL
MOVE-HORIZONTAL

Match border of view: with view: in direction:

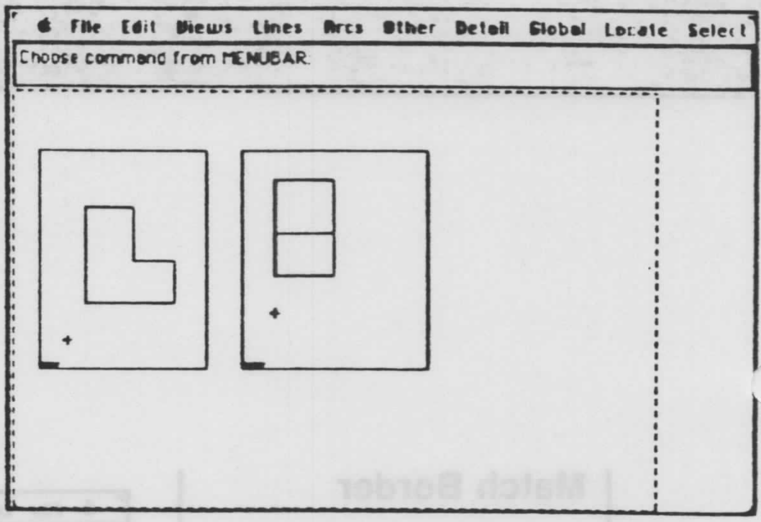
Delete View

The "Delete View" command allows you to delete an entire selected view. When you delete a view, all of the geometry and annotation in that view will be lost. The Workview can not be deleted.

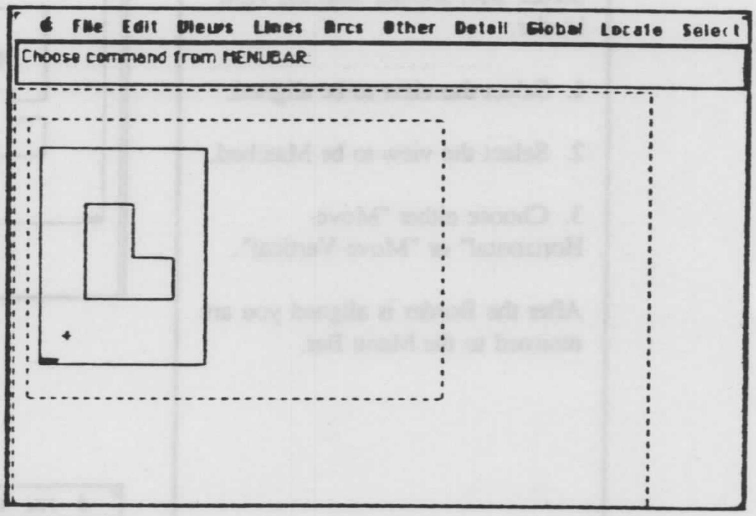
1. Select view to be Deleted.

2. Choose "Enter" to Delete or "Return" to cancel.

In either case you are returned to the Menu Bar.



Choose command from MENUBAR.



Choose command from MENUBAR.

Merge View

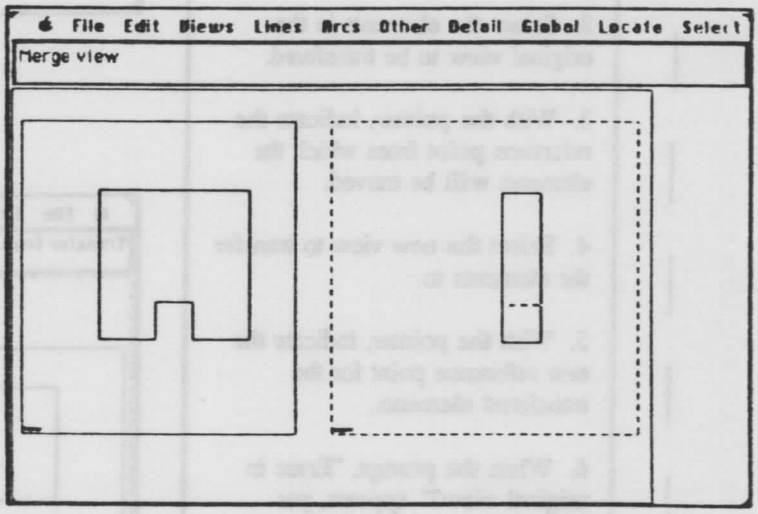
This command allows you to merge two views of a drawing into one.

1. Choose "Merge View".

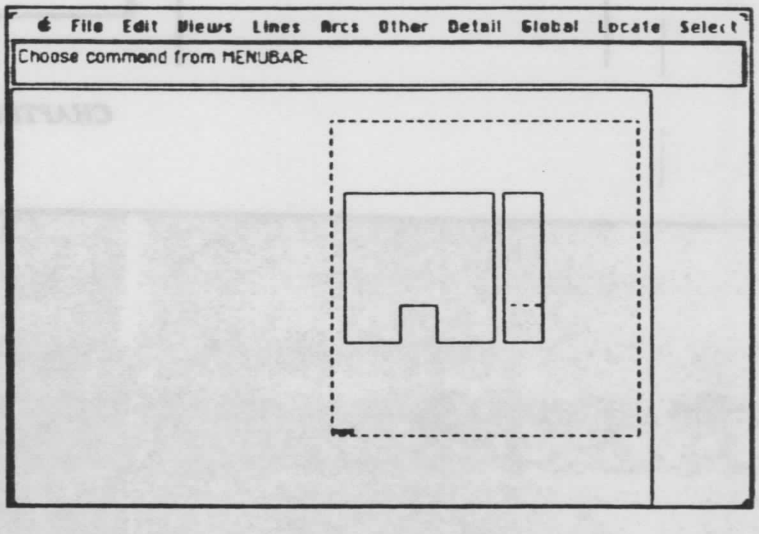
2. Indicate a point in the view to be merged into an existing view.

3. Indicate the point in the existing view where the point from the merged view will be placed.

4. After merging, the merged view will be deleted (see "Delete View") and you will be returned to the Menu Bar.


File Edit Views Lines Arcs Other Detail Global Locate Select

Merge view


File Edit Views Lines Arcs Other Detail Global Locate Select

Choose command from MENUBAR:

Transfer View

This command allows you to transfer geometry from one view to another. With this command, you also have the option of deleting the geometry in the original view, or leaving it to work with later. To transfer geometry from one view to another:

1. Select the view to transfer from. (The prompt," Transfer from view" will appear).

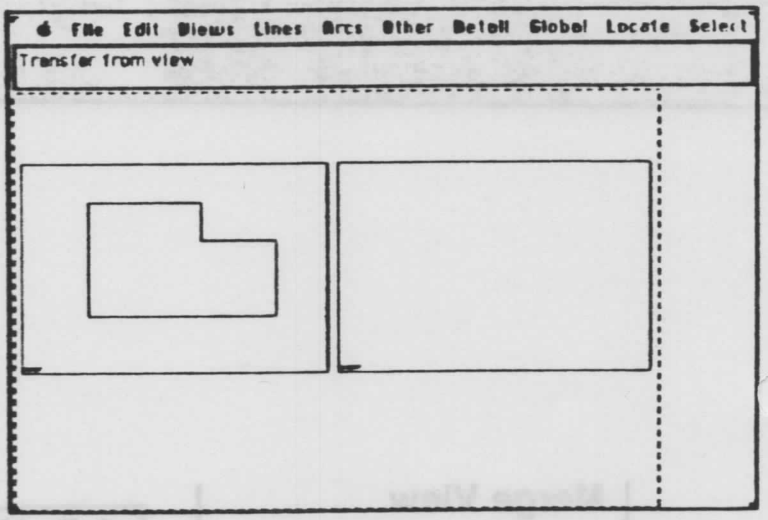
2. Select the elements in the original view to be transfered.

3. With the pointer, indicate the reference point from which the elements will be moved.

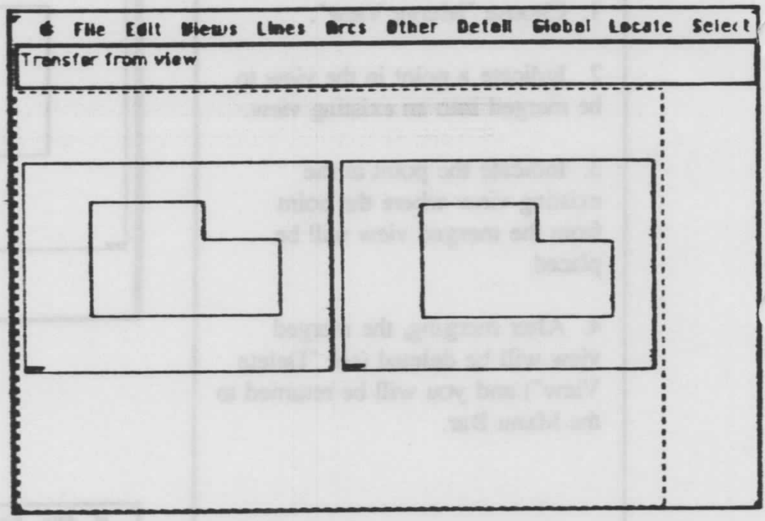
4. Select the new view to transfer the elements to.

5. With the pointer, indicate the new reference point for the transfered elements.

6. When the prompt, "Erase in original view?" appears, yes ("Enter") if you would like the selected elements in the original view to be erased, or no ("Return") if you would like them to remain unchanged.



Transfer from view



Transfer from view

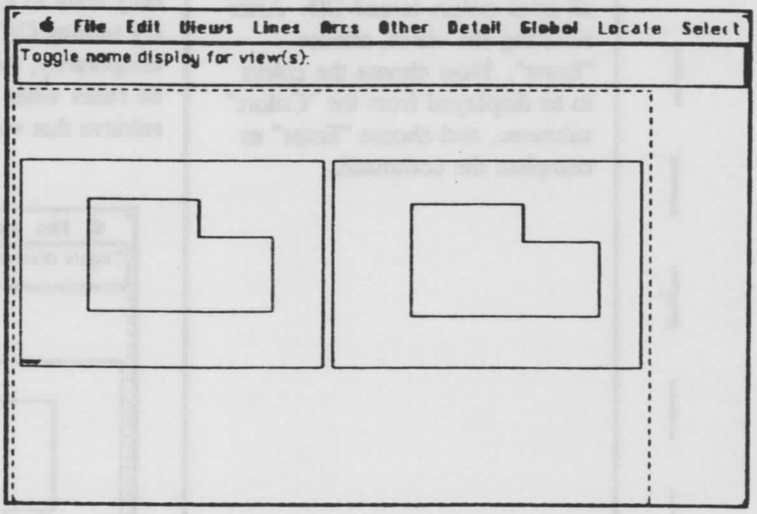
List View

When you select this command, the system will automatically display the values of:

1. Border coordinates relative to the drawing.
2. Origin coordinates relative to the drawing.
3. Trap Value for the selected view.

Name On-Off

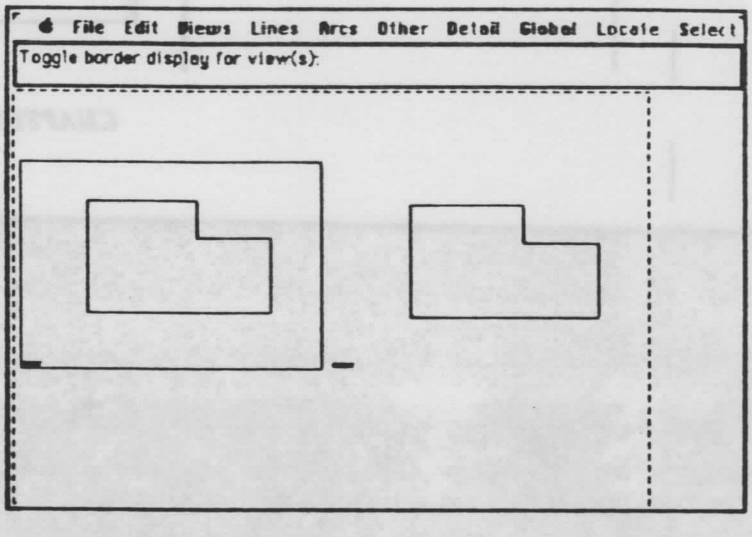
This command allows you to make a view name visible or invisible in a selected view. Choose this command and select the views you wish to toggle (i.e. If On it will turn Off and if Off it will turn On).


File Edit Views Lines Arcs Other Detail Global Locate Select

Toggle name display for view(s):

Border On-Off

This command allows you to make a view border visible or invisible in a selected view. Choose this command and select the views you wish to toggle (i.e. If On it will turn Off and if Off it will turn On).


File Edit Views Lines Arcs Other Detail Global Locate Select

Toggle border display for view(s):

Layers On-Off

This command allows you to specify which layers are visible in selected views, and will leave all others turned Off. After selecting the views, choose "Enter", then enter the range of layers to be displayed.

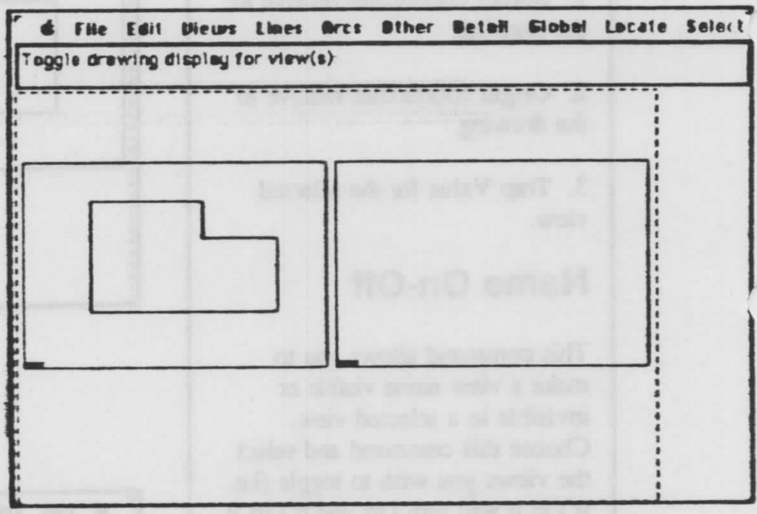
Colors On-Off

This command allows you to specify which colors are visible in the selected view(s), and will leave all other colors turned Off. After selecting the views, choose "Enter". Now choose the colors to be displayed from the "Colors" submenu, and choose "Enter" to complete the command.

View On-Off

This command allows you to turn On or Off the visibility of an entire drawing within specific views. Choose this command and select the views you wish to toggle (i.e. If On it will turn Off and if Off it will turn On). This command does not affect which layers or colors are visible or unblank any blanked elements.

"View On-Off" might be useful when you have three views and only want to look at two of them. By turning Off the third view temporarily, the redraw speed will be faster when you want to retrieve that view.



Toggle drawing display for view(s)

Detail View

This command allows you to take a portion of an existing view and use it to create a detailed view.

1. Select a view with the elements to be detailed.

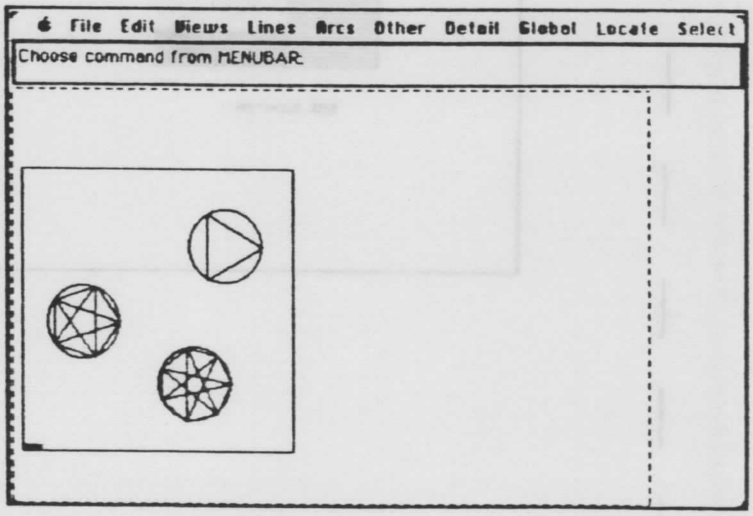
2. Enter the desired magnification factor (e.g. a factor of 2 would make the detailed view scale twice as big).

3. With the pointer, define the portion of the existing view to be detailed.

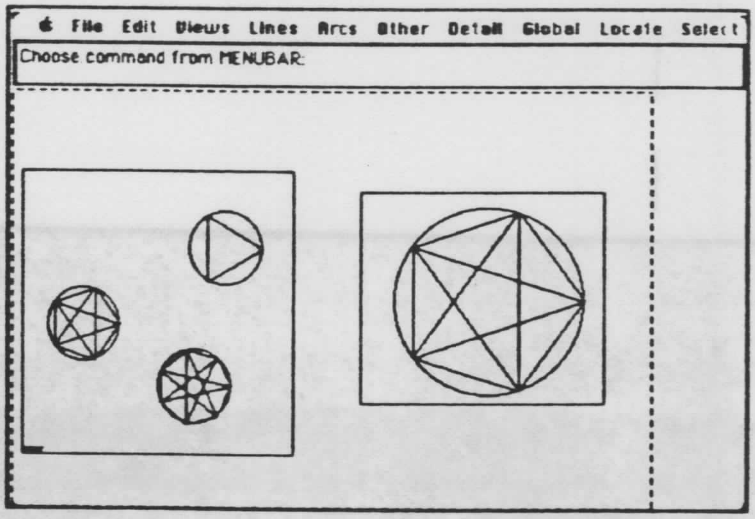
4. With the pointer, select a location for the center of the detail view.

5. Select the elements in the existing view to be detailed.

If you Choose "All" from the "Select" menu when using the "Detail View" command, then the new view may have more elements than needed.

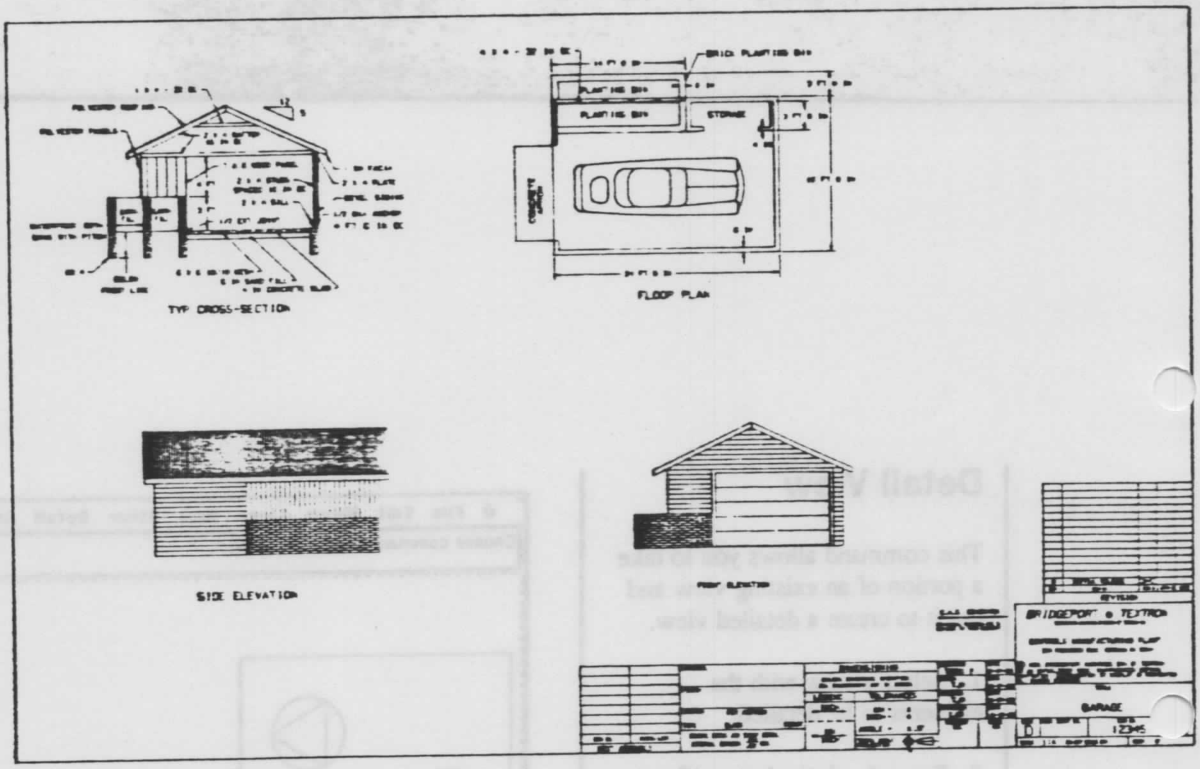


Choose command from MENUBAR:



Choose command from MENUBAR:

Chapter 3



Drawing With EZ-Draft

Drawing with EZ-Draft

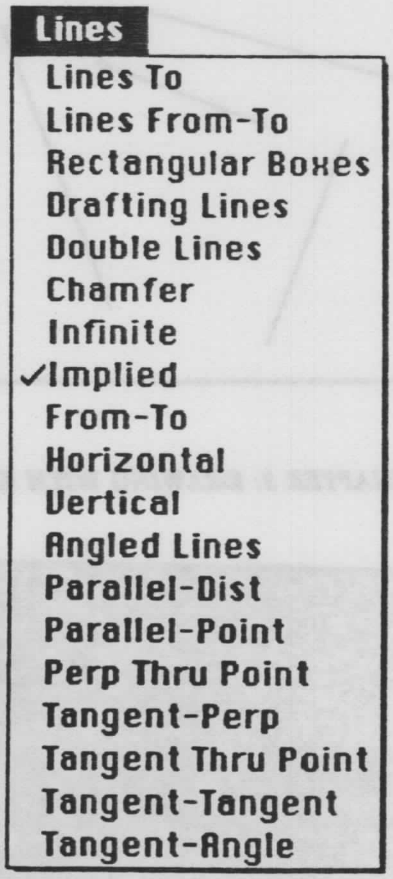
Introduction

This chapter explains the menus and commands which allow you to create geometry with the EZ-Draft system. Each procedure is explained step-by-step, from entering single points and lines to creating complex geometry and symbols. You will quickly learn the menus and commands by following the procedures for creating geometric elements and copying the examples given.

The first piece of geometry created in a view will need an initial point location or reference point defined. This can be digitized or can be entered using Explicit point description (see Chapter 1 Learning EZ-Draft). Subsequent geometry can then be entered using Explicit or Implicit point description or Angle-cross control.

Lines Menu

There are 19 choices in the "Lines" menu, the first 6 choices ("Lines To", "Lines From-To", "Rectangular Boxes", "Drafting Lines", Double Lines" and "Chamfer") can be executed independently of the rest of the commands. "Infinite", "Implied" and "From-To" are mode settings for the remaining 10 choices, with the active mode checkmarked. The "Lines" menu offers 34 different methods of creating lines, the ability to create rectangular boxes and chamfer corners.



Lines To

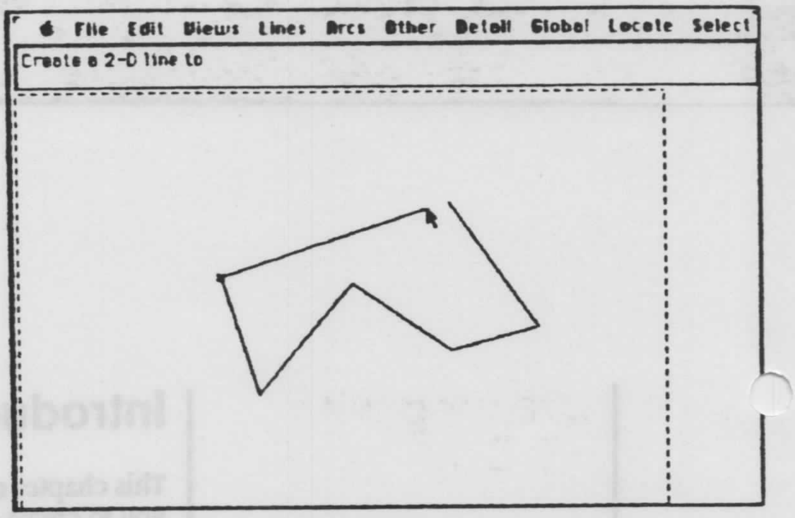
The "Lines To" command allows you to create lines in a continuous or "connected" manner. Actually the lines only appear to be "connected" because they have a common endpoint at the same location.

1. To create the first line, enter the point location of the desired starting point.

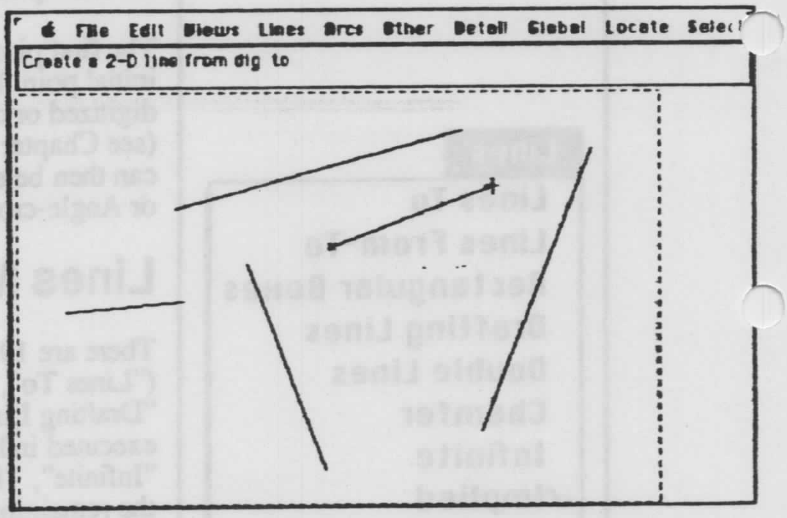
2. Next, enter the point location of the desired ending point of the first line. This is sufficient information to create the first line.

3. To create the second line, enter the ending point, and the line is created. The ending point of the first line is automatically taken to be the starting point of the second line, etc.

You can create lines in this continuous manner until another command is chosen from the Menu Bar or by choosing "Return". Each line created is on the current Layer, uses the current Color, is displayed using the current Curve-font, and has the current Group-name (if one has been set). Note that this is a very efficient way to create a single line of only one line is desired. To create lines which are not connected, see the "Lines From-To" command.



Create a 2-D line to



Create a 2-D line from dig to

Lines From-To

This command allows you to create "unconnected" single lines. To create the single lines that you need:

1. Enter the point location of the desired starting point of the line.

2. Enter the point location of the desired ending point of the line.

3. After entering your second point location, the line is then created between the starting and ending points that you have specified.

The new line is on the current Layer, uses the current Color is displayed with the current Curve-font, and has the current Group-name (if one has been set). You can create lines in this manner until choosing another command from the Menu Bar. You are automatically exited by choosing another command.

Rectangular Boxes

This command allows you to create Rectangular Boxes with square or rounded corners when given two diagonal endpoints. The command actually creates four separate lines in the shape of a box, or four separate arcs and four separate lines in the case of a rounded rectangle. These lines can be used like all other lines for selecting purposes. The radius can be set to a given value or as a proportion of the box width or height. If the radius is a given value (Set-Radius) then the box entered must be at least twice the corner radius or the geometry will not be created. If a proportional radius is given, them the radius is determined as a proportion of the smaller of the width or height.

To create a Rectangular Box:

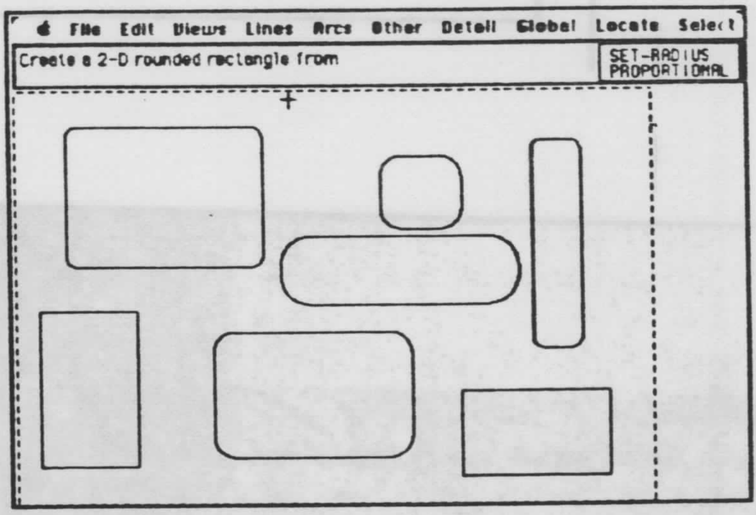
1. Choose "Rectangular Boxes".

2. Enter a radius for the ronded corners (0 gives you a square box).

3. Enter the first endpoint of the box as a point location.

4. Enter the second endpoint of the box as a point location.

This is sufficient information to create the elements of the box as shown. If you choose "Return" at any time to complete the point location of either point, the command is terminated. You will then be returned to the Menu Bar.


File Edit Views Lines Arcs Other Detail Global Locate Select
SET-RADIUS
PROPORTIONAL

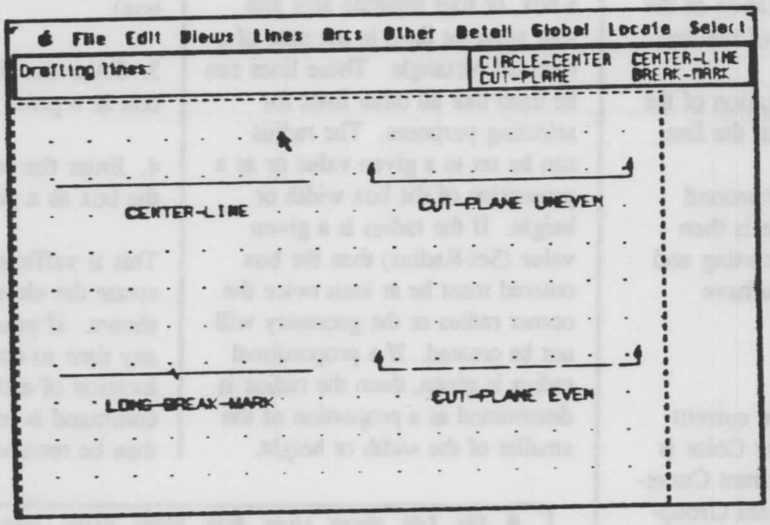
Create a 2-D rounded rectangle from

Drafting Lines

With this command, you can create circle center lines, convert line(s) to center lines, make cutting plane lines, or create a long break mark in a line. When you choose the "Drafting Lines" command, the submenu in the screen below will appear.

Center-Line

This command allows you to convert a line to a Center-Line. This is done by removing segments of the line, creating spaces in the appropriate locations. To convert a line to a Center-Line, select the line to be converted. If you wish to exit the command, choose "Return" or another command.



Cut-Plane (uneven)

This command allows you to convert a line into a Cutting-Plane line with uneven spacing. Segments of the line are removed to create spaces in the appropriate locations and arrows are added, indicating the direction of the section.

Cut-Plane (even)

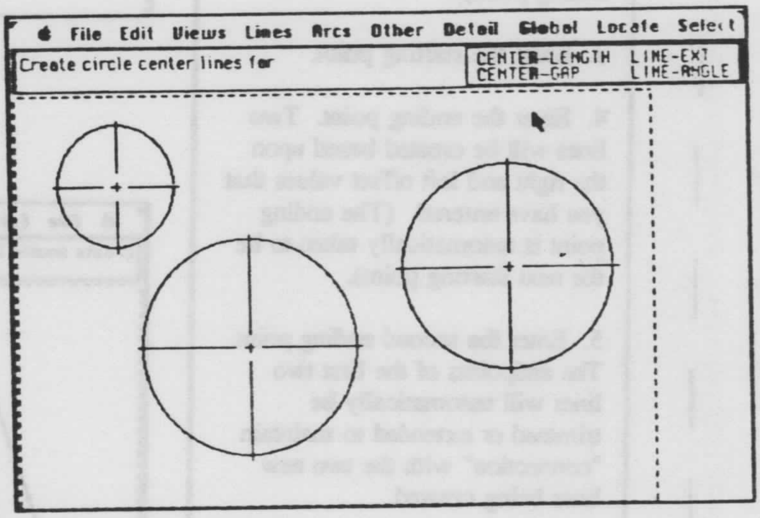
This command allows you to convert a line into a Cutting-Plane line with even spacing. Segments of the line are removed to create spaces in the appropriate locations and arrows are added, indicating the direction of the section.

Break-Mark

This command allows you to place a Break-Mark on a selected line. Simply select the line, then indicate the location for the Break-Mark with the pointer. The Break-Mark will then appear at the position you have indicated.

Circle-Center

This command allows you to create a point ("+") at the center of an arc or circle, and four lines (two horizontal, two vertical). These lines can be used to dimension the location of a circle center. You can also dimension a circle center without a circle center mark. A circle center mark can be adjusted until it has the desired appearance in the drawing. The following commands allow you to make these adjustments.


File Edit Views Lines Arcs Other Detail Global Locate Select
CENTER-LENGTH
CENTER-GAP
LINE-EXT
LINE-ANGLE

Create circle center lines for

Center-Length

This command allows you to set the length of the two lines that make the circle center mark. This mark appears as a "+" onscreen.

Center-Gap

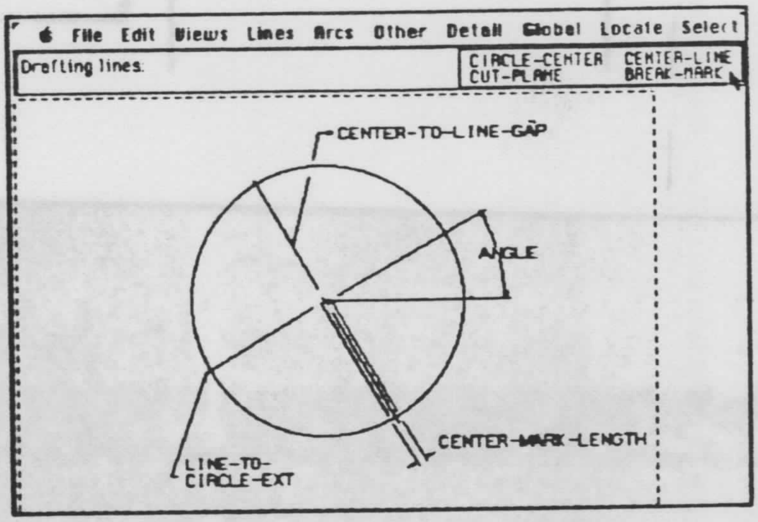
This command allows you to set the gap distance from the circle center mark to the center lines.

Line-Ext

This command allows you to set the distance for the Extension of the circle center lines outside the arc or circle.

Line-Angle

This command allows you to set the angle for the circle center mark (+) and the center lines.



Drafting lines.

Double Lines

This command allows you to create two continuous "connected" parallel lines. Any design application which has walls can use this command to create lines more efficiently than by using other lines commands.

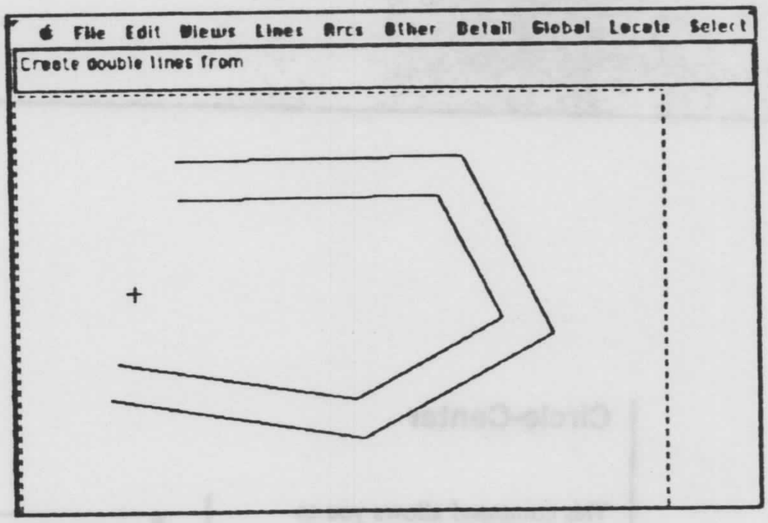
1. Enter a value to be used as the right hand offset.

2. Enter a value to be used for the left hand offset (right and left are determined as if traveling along a line from the starting point to the ending point).

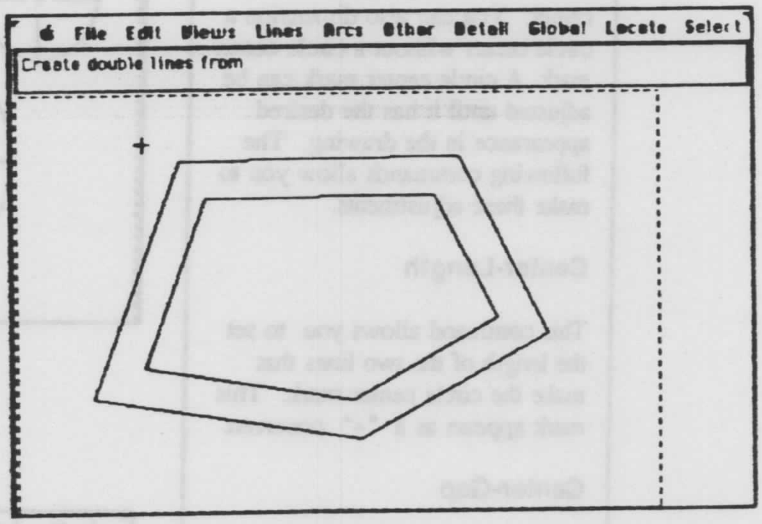
3. Enter the starting point.

4. Enter the ending point. Two lines will be created based upon the right and left offset values that you have entered. (The ending point is automatically taken to be the next starting point).

5. Enter the second ending point. The endpoints of the first two lines will automatically be trimmed or extended to maintain "connection" with the two new lines being created.



Create double lines from



Create double lines from

The "Double Lines" command automatically creates a point at the first entered point. This point is temporary and is erased after the command is exited. You can "close" your wall by selecting this point. When you select this point, both the first two and the last two lines created will automatically be trimmed or extended.

Each of the lines is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). You can continue to create Double Lines in this manner until you choose another command or "Return".

Chamfer

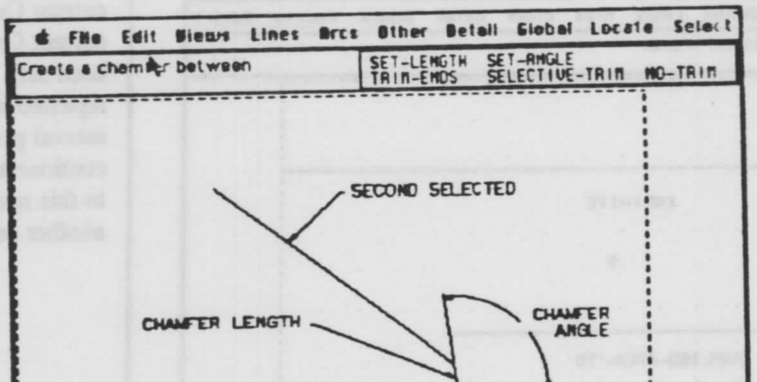
This command allows you to chamfer or "break the corner" where two lines meet. There are two setting options and three trim options with the "Chamfer" command:

1. **Set-Length** to enter a specified Length for subsequent chamfer lines

2. **Set-Angle** to enter the Angle with respect to the first selected line that determines subsequent chamfer lines.

3. Three trim options:

a. **Trim-Ext-Ends**
b. **Selective-Trim**
c. **No-Trim**



File Edit Views Lines Arcs Other Detail Global Locate Select
SET-LENGTH SET-ANGLE
TRIM-ENDS SELECTIVE-TRIM NO-TRIM
SECOND SELECTED
CHAMFER LENGTH
CHAMFER ANGLE
FIRST SELECTED

Create a chamfer between

To use the "Chamfer" command, select two lines with the pointer. These two lines must not be parallel in order to successfully chamfer a corner. The chamfer line is determined by the angle between the two selected [illegible] their point of intersection [illegible] the Chamfer Length and [illegible] Angle settings.

Chamfer Length

The chamfer Length is the actual length of the Chamfer line that is being created.

Chamfer Angle

The Chamfer Angle is the angle with respect to the first selected line.

1. Select the first line. The midpoints of the four possible chamfer lines are displayed onscreen with small x's.

2. Next, select the midpoint of the chamfer line you intended.

The line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set).

If the **Trim-Ext** option is set, the two selected lines are automatically trimmed or extended to the appropriate endpoint of the chamfer line.

If the **Selective-Trim** option is set, you are given the opportunity to select none, one, or both of the selected lines to be trimmed or extended.

If the **No-Trim** option is set, then the selected lines are left unchanged by the "Chamfer" command. The "Chamfer" command is unaffected by the "Infinite", "Implied" or "From-To" modes in the "Lines" menu.

Set-Length

This command sets the length of subsequent chamfer lines (see Number Entry in Chapter 1 for more details).

Set-Angle

This command sets the Chamfer Angle used by subsequent chamfer operations. This angle is used with respect to the first selected line to determine the chamfer line (see Angle Entry Menu in Chapter 1 for more details).

Trim/Ext-Ends

This command sets the trim option to automatically trim or extend the selected lines that determine a chamfer line to the appropriate chamfer line endpoint.

Selective-Trim

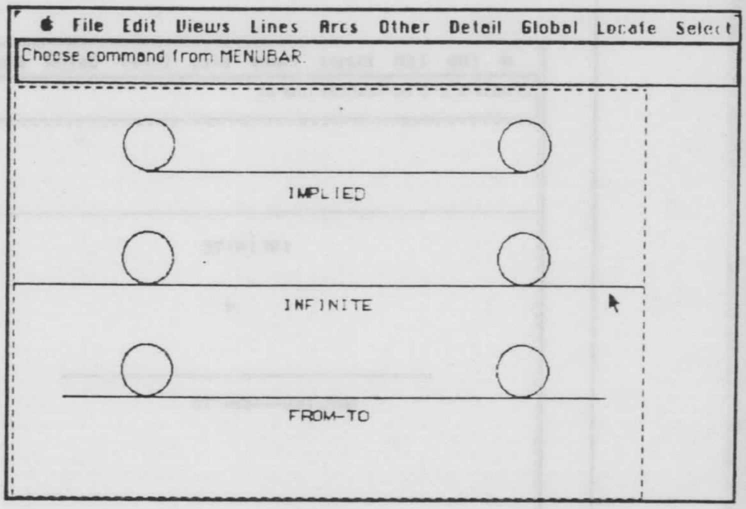
This command sets the trim option to allow you to select none, one, or both of the selected lines that determine a chamfer line to be trimmed or extended to the appropriate chamfer line endpoint.

No-Trim

This command turns Off the trim option.

Infinite Mode

This line mode setting causes lines that you have created to start at the near boundary of the current view and end at its far boundary. This mode is useful to rapidly create construction lines that can then be used to define points by intersections. The lines can also be trimmed with the "Edit Lines" or "Trim End" commands in the "Edit" menu (see "Making Changes and Finishing Up" Chapter 4).



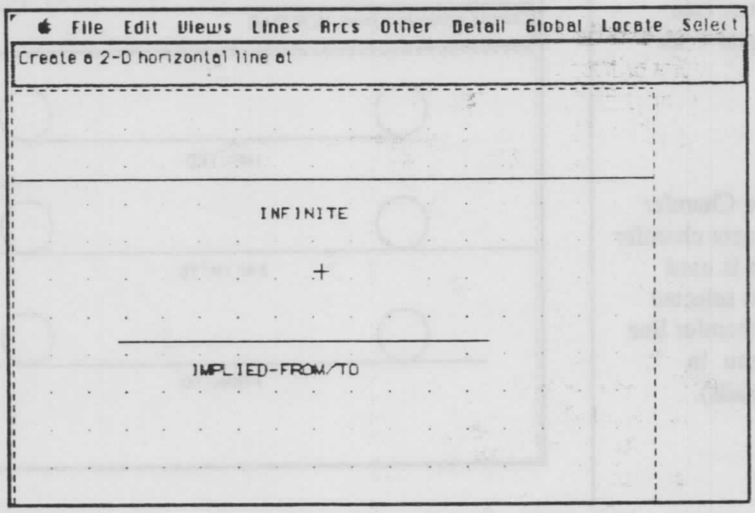
Choose command from MENUBAR.

Implied Mode

This line mode setting causes lines that you have created to be the "most likely desired" length and location (See the figure on the previous page). For example, a line created with the "Tangent Thru Point" command in the "Implied" mode would start at the entered point and end at the tangent point (see each command description for the implied line).

From-To Mode

This line mode setting causes lines that you have created to begin at a specified starting (From) point and end at a specified ending (To) point. If the command has already established the line's angle and/or location, then the From point is determined by the perpendicular drop from the entered point to the established line. Similarly, if the command has already established the line's angle and/or location after knowing the From point, then the To point is determined by the perpendicular drop from the entered point to the established line.



Create a 2-D horizontal line at

Horizontal

This command creates horizontal lines, and is also affected by the line mode settings. The "From-To" mode is the same as the "Implied" mode. The "From-To" mode uses the first entered (From) point as the starting point for the line. The X coordinate of the second entered point is used to determine the ending (To) point (See the figure on this page).

The "Infinite " mode creates a horizontal line at the Y coordinate of the first and only entered point. The line extends from the left view boundary to the right view boundary.

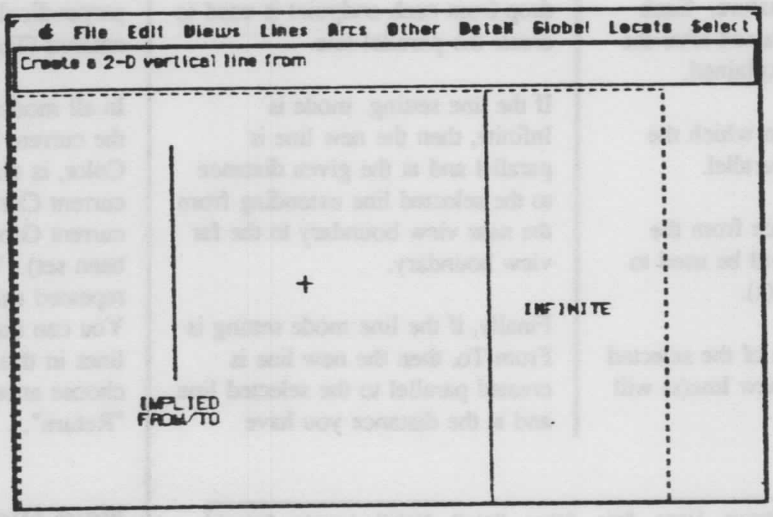
In all modes, the line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). The process can be repeated after you finish the second point description. You can continue to create horizontal lines in this manner until you choose another command or "Return".

Vertical

This command allows you to create vertical lines (See the figure on this page), and is also affected by the line mode settings. The From-To mode is the same as the "Implied" mode. The From-To mode uses the first entered (From) point as the starting point for the line. The Y coordinate of the second entered point is used to determine the ending (To) point.

The "Infinite" mode creates a vertical line at the X coordinate of the first and only entered point. The line extends from the bottom view boundary to the top view boundary.

In all modes, the line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). The process can be repeated after you finish the second point description. You can continue to create vertical lines in this manner until you choose another command or "Return".


File Edit Views Lines Arcs Other Detail Global Locate Select
IMPLIED
FROM/TO
INFINITE

Create a 2-D vertical line from

Angled Lines

This command allows you to create Angled Lines, and is also affected by the line mode settings. The "From-To" mode is the same as the "Implied" mode. First, enter the desired angle (see "Angle Entry" menu in Chapter 1). The "From-To" mode uses the first entered (From) point as the starting point for the line. The perpendicular drop from the second entered point is used to determine the ending (To) point.

The "Infinite" mode creates an angled line that passes through the first and only entered point at the specified angle. The line extends from the near view boundary to the far view boundary.

In all modes, the line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). The process can be repeated after you finish the second point description. You can continue to create angled lines in this manner until you choose another command or "Return".

Parallel Dist

This command allows you to create lines that are parallel to an existing line at a given distance. This command is unique because after its first execution, you are offered several options to create more lines of this nature. Each option will be explained after the first execution is explained.

1. Select the line to which the new lines will be parallel.

2. Enter the distance from the selected line that will be used to create the next line(s).

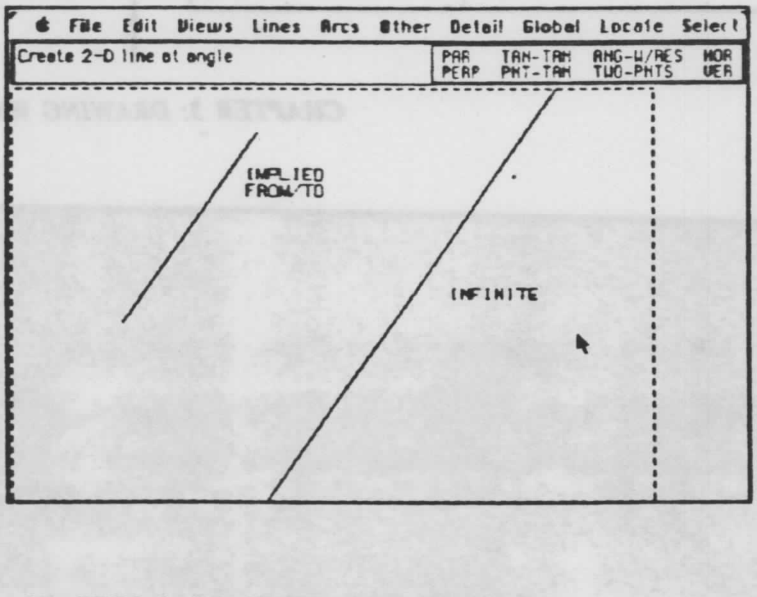
3. Choose the side of the selected line on which the new line(s) will be created.

If the line mode setting is Implied, a line is now created parallel to the selected line at the distance you have entered. The length of the new line is equal to the selected line. A perpendicular drop from each endpoint is used to create the parallel line.

If the line setting mode is Infinite, then the new line is parallel and at the given distance to the selected line extending from the near view boundary to the far view boundary.

Finally, if the line mode setting is From-To, then the new line is created parallel to the selected line, and at the distance you have entered. Its starting point is determined by taking the perpendicular drop from the entered (From) point to the new line. Its ending (To) point is determined by taking the perpendicular drop from the entered (To) point to the new line.

In all modes, the line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). The process can be repeated using the submenus. You can continue to create parallel lines in this manner until you choose another command or "Return".


File Edit Views Lines Arcs Other Detail Global Locate Select
PAR TAN-TAN ANG-W/RES HOR
PERP PNT-TAN TWO-PNTS VER
IMPLIED
FROM/TO
INFINITE

Create 2-D line at angle

New Distance

This selection allows you to create another line parallel to the selected line. Enter the new distance from the selected line. This distance will determine the new line. The new line is then created in the same manner (and on the same side) as the first parallel line.

Inc-Distance

This selection allows you to create a new line parallel to the last line created. Enter an incremental distance from the last parallel line. This will determine the new line.

Step-And-Repeat

This selection allows you to create a line parallel to the last line created. You will enter a step distance for the new line(s), and the number of times to repeat the step distance. After this command, other selections such as New-Distance can be used. As before, the new lines are created in the same manner as the first parallel line.

New-Base-Line

This command allows you to choose a new line as the reference line for creating parallel lines at the last entered distance.

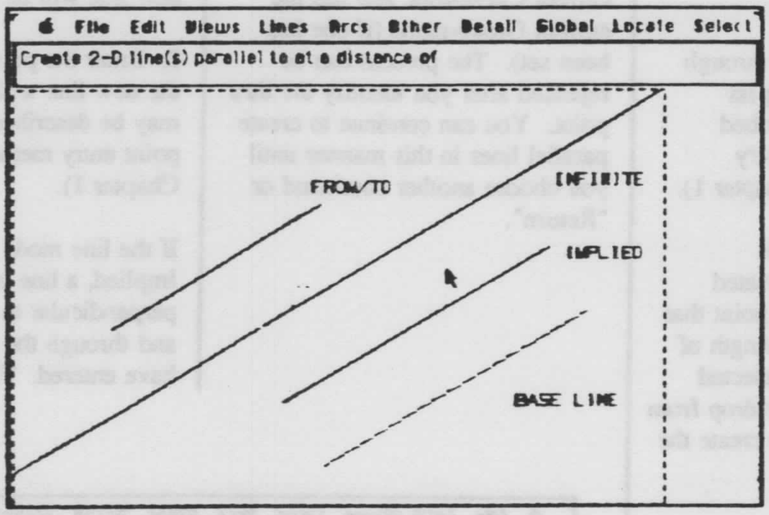
1. Choose New-Base-Line

2. With the pointer, indicate the new reference line.

3. Indicate the side on which the parallel line will be created.

New-Side

This selection allows you to indicate a new side on which to create another parallel line at the last distance. This command is useful when creating a line on each side of an existing line. The first line will be created as above, and only the side needs to be indicated to create the second line. The new line is created in the same manner as the first parallel line.


File Edit Views Lines Arcs Other Detail Global Locate Select
FROM/TO
INFINITE
IMPLIED
BASE LINE

Create 2-D line(s) parallel to at a distance of

Parallel-Point

This command allows you to create lines parallel to an existing line through a point that you have entered.

1. Select the line to which the new line will be parallel.

2. Then, enter the point through which the new line will pass (This point may be described using any of the point entry methods described in Chapter 1).

If the line mode setting is Implied, a line is now created parallel and through the point that you have entered. The length of the line is equal to the selected line, and a perpendicular drop from each endpoint is used to create the parallel line.

If the line mode setting is Infinite, then the new line is parallel and passes through the given point. It extends from the near view boundary to the far view boundary.

Finally, if the line mode setting is From-To, then the new line is created parallel and through the given point. Its starting point is determined by taking the perpendicular drop from the entered (From) point to the new line. Its ending point is determined by taking the perpendicular drop from the entered (To) point to the new line.

In all modes, the line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). The process can be repeated after you identify the thru point. You can continue to create parallel lines in this manner until you choose another command or "Return".

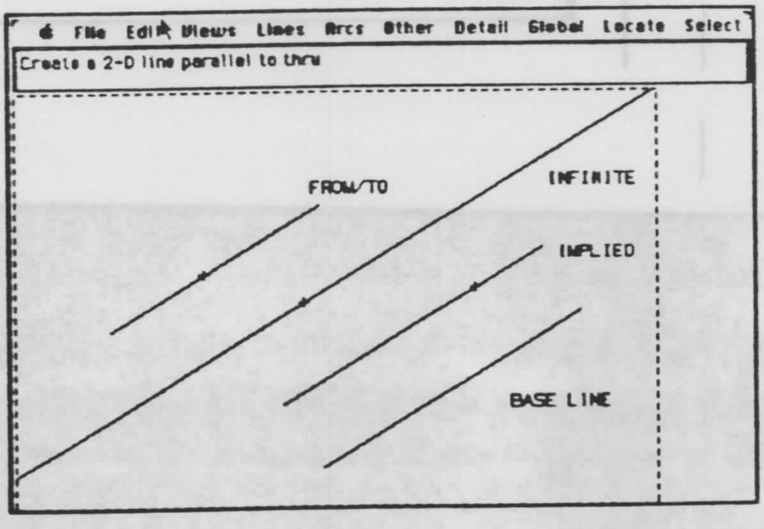
Perp-Thru-Point

This command allows you to create a line perpendicular to an existing line and passing through a point that you have entered.

1. Select the line to which the new line will be perpendicular.

2. Enter the point through which the new line will pass (this point may be described using any of the point entry methods described in Chapter 1).

If the line mode setting is Implied, a line is created perpendicular to the selected line and through the point that you have entered. The starting point


File Edit Views Lines Arcs Other Detail Global Locate Select
FROM/TO
INFINITE
IMPLIED
BASE LINE

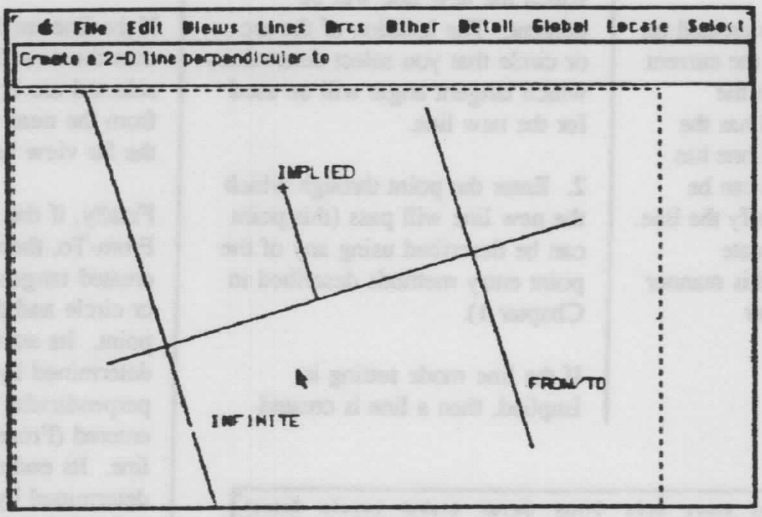
Create a 2-D line parallel to thru

is the point that you have entered. The ending point is the point where the perpendicular line intersects the selected line.

If the line mode setting is Infinite, then the new line is perpendicular to the selected line and passes through the given point, extending from the near view boundary to the far view boundary.

Finally, if the line mode setting is From-To, then the new line is created perpendicular to the selected line and through the given point. Its starting point is determined by taking the perpendicular drop from the entered (From) point, to the new line. Its ending point is determined by taking the perpendicular drop from the entered (To) point to the new line.

In all modes, the line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). The process can be repeated after you identify the thru point. You can continue to create perpendicular lines in this manner until you choose another command or "Return".


File Edit Views Lines Arcs Other Detail Global Locate Select
IMPLIED
FROM/TO
INFINITE

Create a 2-D line perpendicular to

Tangent-Perp

This command allows you to create a line tangent to an existing arc/circle and perpendicular to an existing line.

1. Select the arc or circle to which the new line will be tangent. The location of the arc or circle that you select is important, because it determines which of two tangent points will be used for the new line.

2. Select the line to which the new line will be perpendicular.

If the line mode setting is Implied, a line is created tangent to the selected arc or circle and perpendicular to the selected line. The starting point is the tangent point to the arc or circle. The ending point is where the new line intersects the selected line.

If the line mode setting is Infinite, then the new line is tangent to the selected arc or circle and perpendicular to the selected line. It extends from the near view boundary to the far view boundary.

Finally, if the line mode setting is From-To, then the new line is created tangent to the selected arc or circle and perpendicular to the selected line. Its starting point is determined by taking the perpendicular drop from the entered (From) point to the new line. Its ending point is determined by taking the perpendicular drop from the entered (To) point to the new line.

In all modes, the line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). The process can be repeated after you identify the line. You can continue to create perpendicular lines in this manner until you choose another command or "Return".

Tangent-Thru-Point

This command allows you to create a line tangent to an existing arc or circle, passing through a point that you have entered.

1. Select the arc or circle to which the new line will be tangent. The location of the arc or circle that you select determines which tangent angle will be used for the new line.

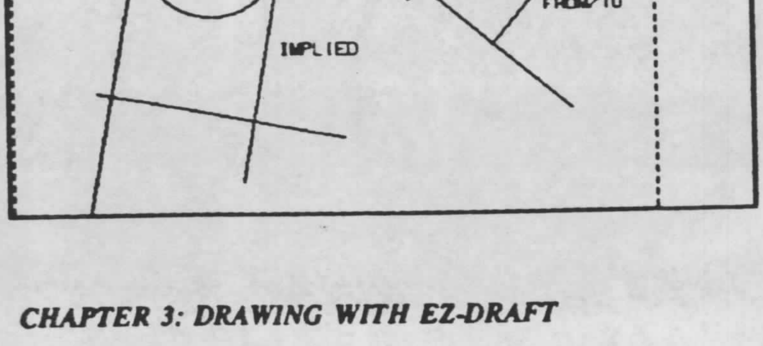
2. Enter the point through which the new line will pass (this point can be described using any of the point entry methods described in Chapter 1).

If the line mode setting is Implied, then a line is created tangent to the selected arc or circle through the point that you have entered. The starting point is the point that you have entered. The ending point is the point tangent to the arc or circle.

If the line mode setting is Infinit then the new line is tangent to the selected arc or circle, extending from the near view boundary to the far view boundary.

Finally, if the line mode setting is From-To, then the new line is created tangent to the selected ar or circle and through the given point. Its starting point is determined by taking the perpendicular drop from the entered (From) point to the new line. Its ending point is determined by taking the perpendicular drop from the entered (To) point to the new line.

In all modes, the line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). The process can be repeated after you identify the point location. You can contin to create perpendicular lines in this manner until you choose another command or "Return".



Create a 2-D line tangent to and perpendicular

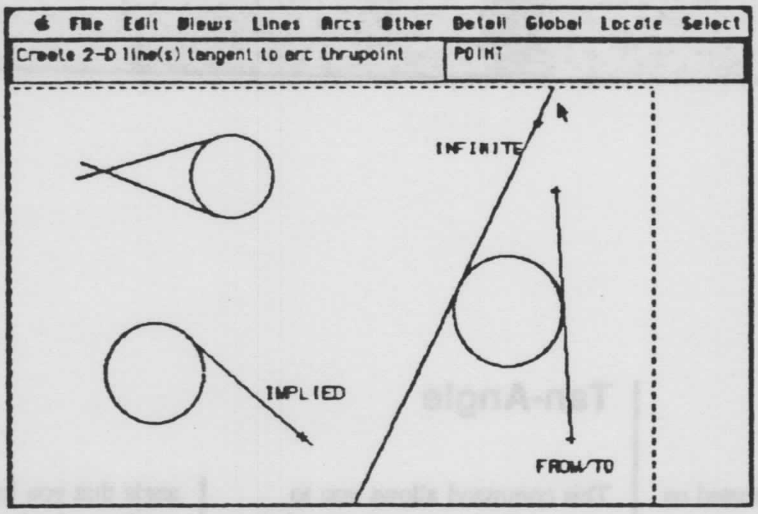
Tangent-Tangent

This command allows you to create a line tangent to two existing arcs or circles. You begin by selecting the two arcs to which the new line will be tangent. The location of the arc that you select is important, because it determines which of four possible tangent angles will be used.

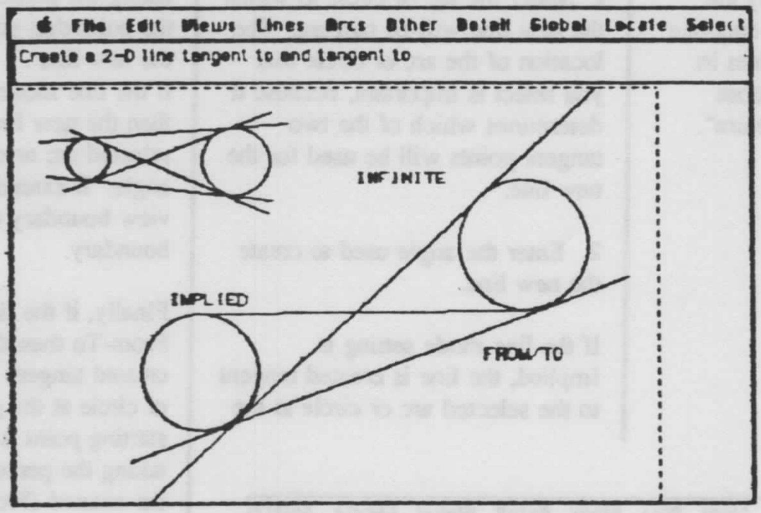
If the line mode setting is Implied, a line will be created tangent to the two arcs, with its starting point at the tangent point of the first selected arc. Its ending point will be at the tangent point of the second selected arc.

If the line mode setting is Infinite, then the new line will be tangent to the two selected arcs, extending from the near view boundary to the far view boundary.

Finally, if the line mode setting is From-To, then the new line will be created tangent to the two selected arcs. Its starting point will be determined by taking the perpendicular drop from the entered (From) point to the new line. Its ending point will be determined by taking the perpendicular drop from the entered (To) point to the new line.


File Edit Views Lines Arcs Other Detail Global Locate Select
INFINITE
IMPLIED
FROM/TO

Create 2-D line(s) tangent to arc thrupoint | POINT


File Edit Views Lines Arcs Other Detail Global Locate Select
INFINITE
IMPLIED
FROM/TO

Create a 2-D line tangent to arc tangent to

In all modes, the line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). The process can be repeated after you identify the point location. You can continue to create perpendicular lines in this manner until you choose another command or "Return".

Tan-Angle

This command allows you to create a line tangent to an existing arc or circle at an angle that you have entered (see "Angle Entry" menu in Chapter 1).

1. Select the arc or circle to which the new line will be tangent. The location of the arc or circle that you select is important, because it determines which of the two tangent points will be used for the new line.

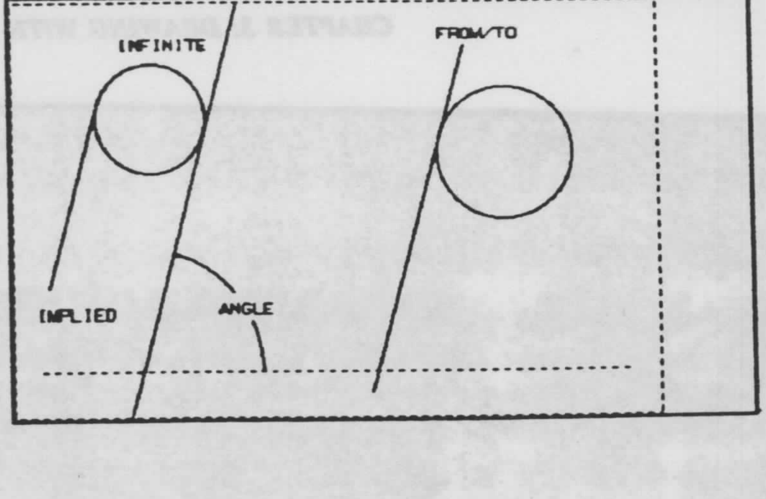
2. Enter the angle used to create the new line.

If the line mode setting is Implied, the line is created tangent to the selected arc or circle at the angle that you have entered. The starting point is the tangent to the arc or circle. The tracking line is active and helps you by showing where the line and its angle start. The ending point is determined by taking the perpendicular drop from the point that you have entered t the new line .

If the line mode setting is Infinite, then the new line is tangent to the selected arc or circle at the entered angle. It extends from the near view boundary to the far view boundary.

Finally, if the line mode setting i From-To then the new line is created tangent to the selected arc or circle at the given angle. Its starting point is determined by taking the perpendicular drop from the entered (From) point to the new line. Its ending point is determined by taking the perpendicular drop from the entered (To) point to the new line.

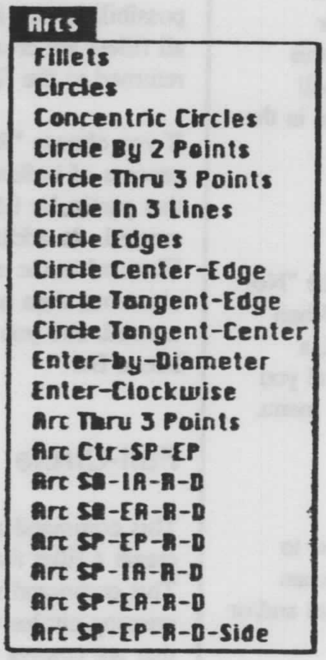
In all modes, the line is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set). The process can be repeated after you identify the point location. You can continu to create perpendicular lines in this manner until you choose another command or "Return".



Create a 2-D line tangent to at an angle of

There are 20 choices offered in the "Arcs" menu, 9 different methods to describe circles (closed arcs) and 8 methods for describing arcs. Information can be entered in "Radius" or "Diameter" mode and "Clockwise" or "Counterclockwise" mode. The "Arcs" menu also offers the ability to fillet selected lines.

In all modes, the geometry created from the "Arcs" menu is created on the current Layer, uses the current Color, is displayed with the current Curve-font, and has the current Group-name (if one has been set).



Fillets

The commands in this menu allow you to select lines and/or arcs to be filleted, using a set fillet radius (See the figure on this page).

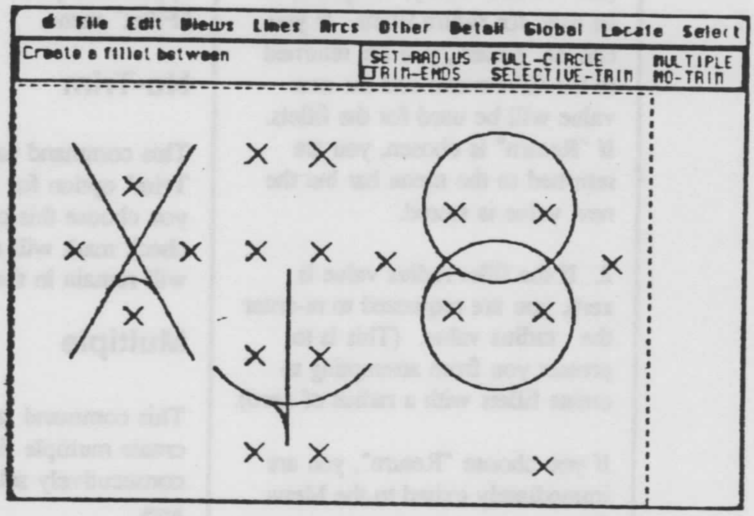
1. Set a fillet radius (See Set Radius on the next page).

2. Select two lines and/or arcs.

3. Indicate the center for the fillet. The possible centers will be indicated onscreen by small x's (if they do not lie off the screen area). In any case, you should digitize near the desired fillet center.

If the Trim/Ext-Ends option is On, then the ends of the lines and/or arcs will automatically be trimmed to the appropriate fillet endpoint.

If the Selective-Trim option is On, you are requested to indicate which (if any) line or arc should be trimmed. If you choose "Return" during Selective-Trim, you are returned to the Menu Bar. If the No-Trim option is On, then the fillet is created, but no trimming or extending is performed. You can continue to create fillets until you exit the command by choosing "Return".


Create a fillet between

Set -Radius

This command allows you to set a fillet radius.

1. Enter a value or implicit radius (see Number Entry, Chapter1) to be used for future fillets. If you choose "Enter", you are returned to the fillet menu and the new value will be used for the fillets. If "Return" is chosen, you are returned to the menu bar but the new value is stored.

2. If the fillet radius value is zero, you are requested to re-enter the radius value. (This is to protect you from attempting to create fillets with a radius of zero).

If you choose "Return", you are immediately exited to the Menu Bar.

Trim/Ext-Ends

This command turns On the "Trim/Ext-Ends" option for fillets. When you choose this command, a check mark will appear and you will remain in the "Fillet" menu.

Selective-Trim

This command turns On the "Selective-Trim" option for fillets. When you choose this command, a check mark will appear and you will remain in the "Fillet" menu.

No-Trim

This command turns On the "No-Trim" option for fillets. When you choose this command, a check mark will appear and you will remain in the "Fillet" menu.

Multiple

This command allows you to create multiple fillets between consecutively selected lines and/or arcs.

1. Select any number of lines and/or arcs between which the fillets will be created. After selecting the lines and/or arcs choose "Enter", then the first two elements that you select are used to indicate the possible fillets.

2. After you indicate the center for that fillet, the second and third selections are used to indicate possible fillets. This process continues until you make your last selection.

If the last selection has an endpoint in common with the first selection, then the last fillet possibilities are indicated. After all fillets are created, you are returned to the fillet menu.

If you choose "Return" during the process of indicating centers, the that particular fillet will not be created. In addition, if you choose "Return" at the end of selecting elements, then no fillets are created, and you are returned to the Menu Bar.

Full-Circle

This command allows you to create a fillet that is a full circle. This command is useful for creating circles with a set radius that are tangent to two elements. Typically, the "Full-Circle" command would be used with the "No-Trim" option.

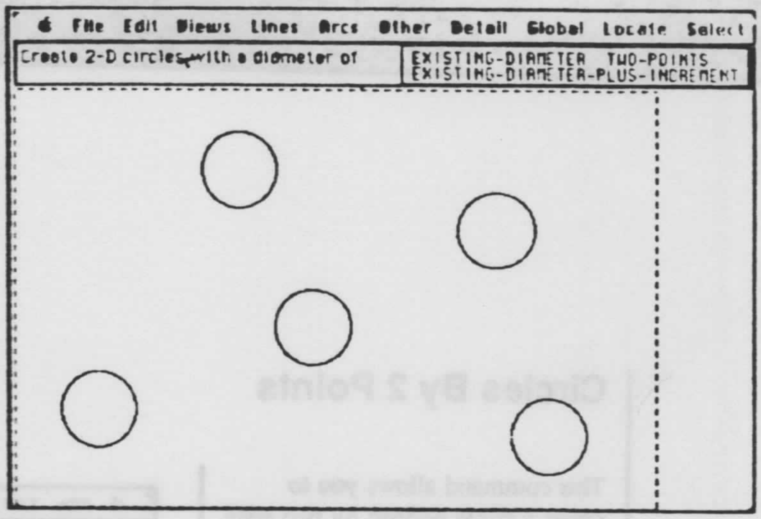
Circles

This command allows you to create circles of a given radius or diameter with various center locations.

1. Enter the radius or diameter for the circles.

2 Next, enter the center location(s) for each circle.

The process can be repeated after you enter the center location. When repeated, enter the new center since this command uses the same radius or diameter as used for the first circle. You can continue to create circles in this manner until you choose another command or "Return".


File Edit Views Lines Arcs Other Detail Global Locate Select
EXISTING-DIAMETER TWO-POINTS
EXISTING-DIAMETER-PLUS-INCREMENT

Create 2-D circles with a diameter of

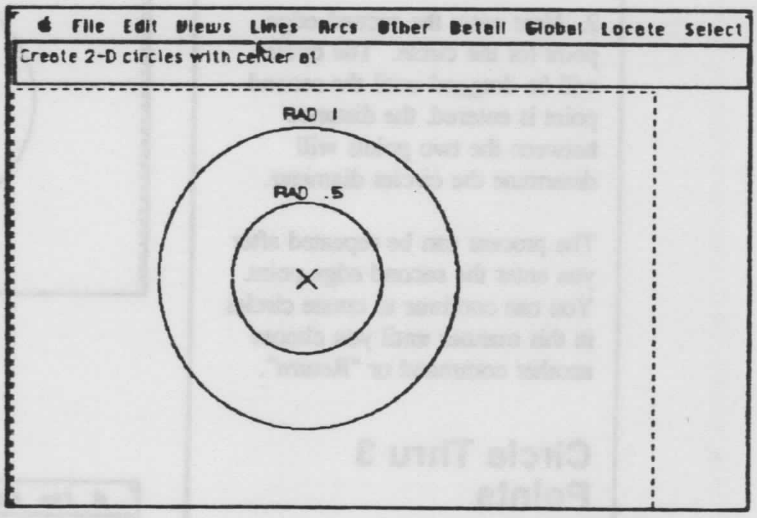
Concentric Circles

This command allows you to create concentric circles defined by a center and a series of radii or diameters.

1. Enter the center to be used for the circles.

2. Enter a radius or diameters for the first circle.

The process can be repeated after you enter the radius or diameter and choose "Enter". You can continue to create circles in this manner until you choose another command or "Return".


File Edit Views Lines Arcs Other Detail Global Locate Select
RAD 1
RAD .5

Create 2-D circles with center at

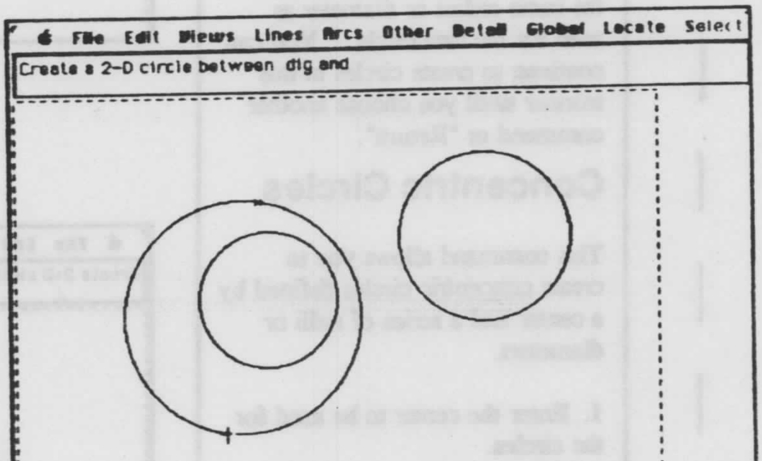
Circles By 2 Points

This command allows you to create a circle defined by two edge points.

1. Enter the first edge point for the circle.

2. Next enter the second edge point for the circle. The circle will be dragged until the second point is entered. the distance between the two points will determine the circles diameter.

The process can be repeated after you enter the second edge point. You can continue to create circles in this manner until you choose another command or "Return".

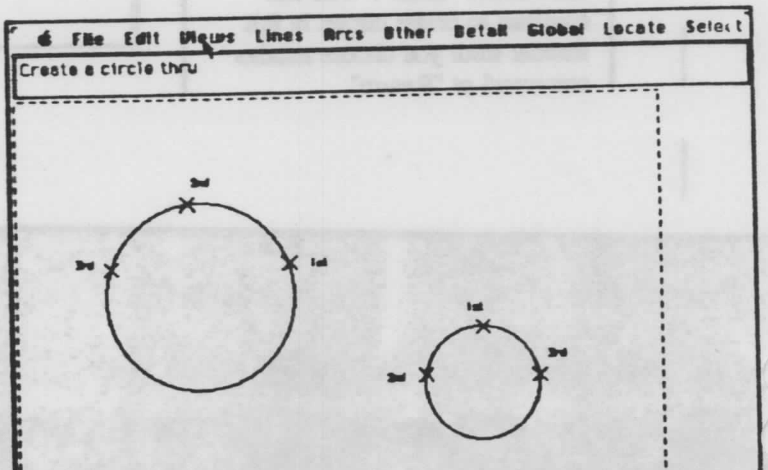


Create a 2-D circle between dig and

Circle Thru 3 Points

This command allows you to create a circle that passes through 3 points.

Enter the 3 points through which the new circle will pass. The process can be repeated after you enter the the third point and choose "Enter". You can continue to create circles in this manner until you choose another command or "Return".

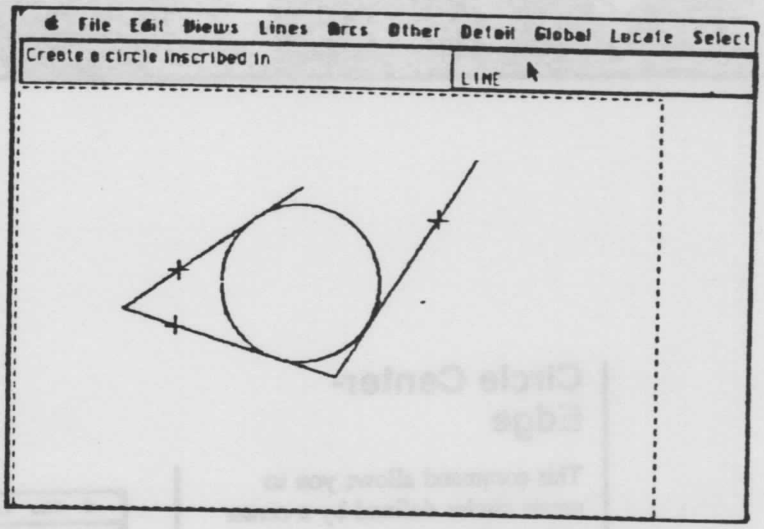


Create a circle thru

Circle In 3 Lines

This command allows you to create a circle inscribed in 3 lines.

Select the three lines in which the circle will be inscribed. The process can be repeated after you enter the third line and choose "Enter". You can continue to create circles in this manner until you choose another command or "Return".



Create a circle inscribed in

LINE

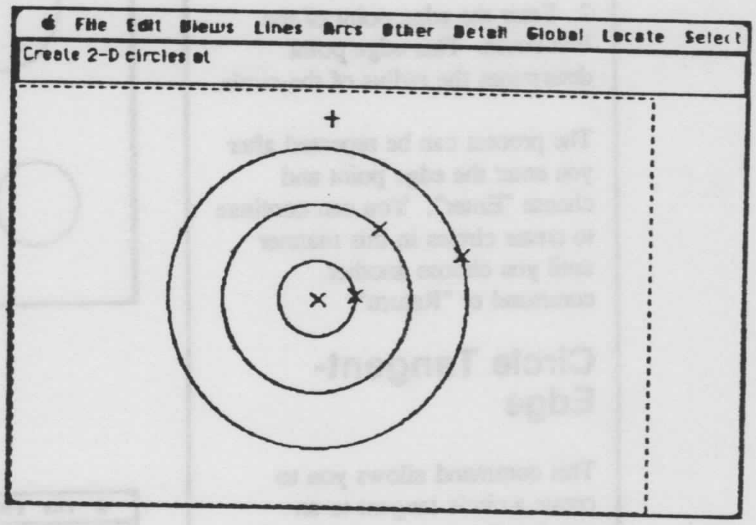
Circle Edges

This command allows you to create concentric circles defined by a center and a series of edge points.

1. Enter the center to be used for the circles.

2. Next, enter an edge point on the circle. This edge point is used to calculate the radius of the circle, thus the circle is created.

The process can be repeated after you enter the edge point and choose "Enter". You can continue to create circles in this manner until you choose another command or "Return".



Create 2-D circles at

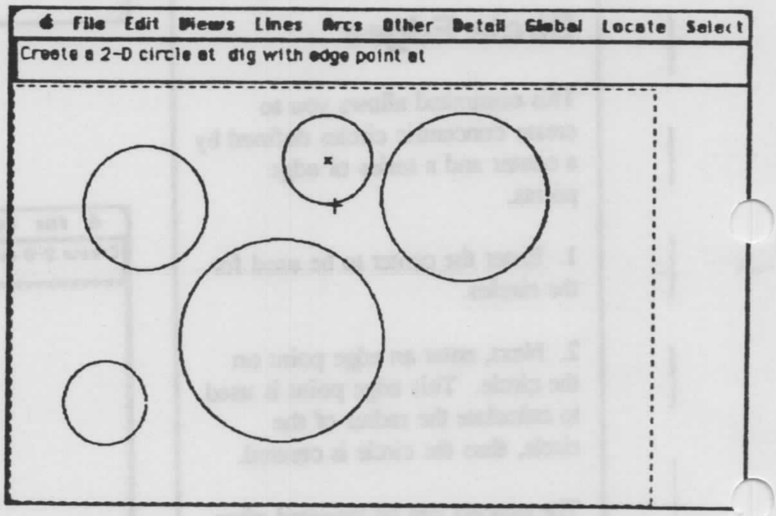
Circle Center-Edge

This command allows you to create circles defined by a center and an edge point.

1. Enter the center of the first circle.

2. Enter the edge point of the first circle. This edge point determines the radius of the circle.

The process can be repeated after you enter the edge point and choose "Enter". You can continue to create circles in this manner until you choose another command or "Return".



Create a 2-D circle at dig with edge point at

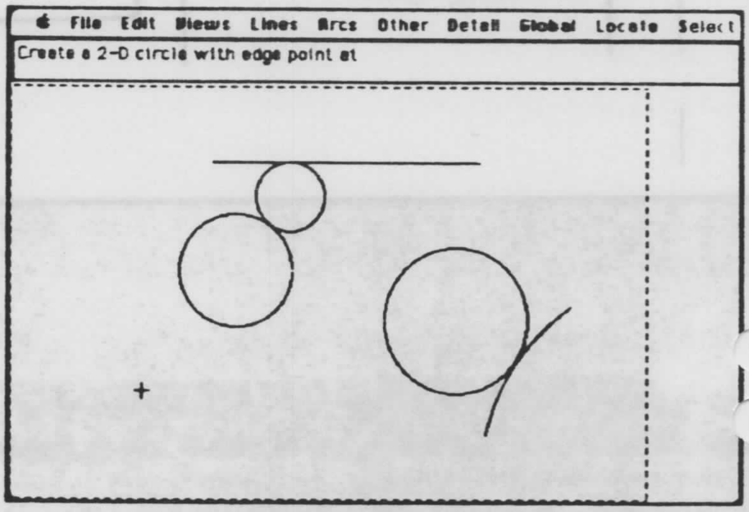
Circle Tangent-Edge

This command allows you to create a circle tangent to an existing line, arc, or circle with an edge point that you have selected.

1. Enter the location of the new circle's edge point.

2. Next, select the element (line, arc, or circle) to which the new circle will be tangent.

The process can be repeated after you select the desired element and choose "Enter". You can continue to create circles in this manner until you choose another command or "Return".



Create a 2-D circle with edge point at

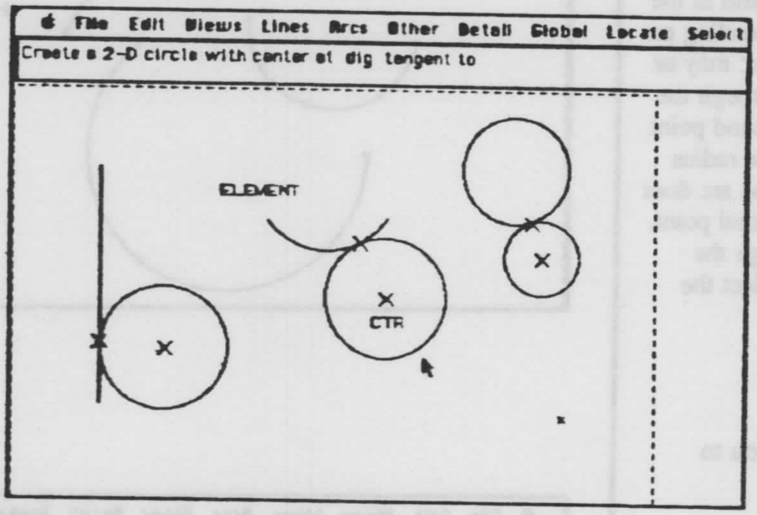
Circle Tangent Center

This command allows you to create a circle tangent to an existing line, arc, or circle with a center that you have entered.

1. First, enter the location of the new circle's center.

2. Next, select the element (line, arc, or circle) to which the new circle will be tangent.

The process can be repeated after you select the desired element and choose "Enter". You can continue to create circles in this manner until you choose another command or "Return".



Create a 2-D circle with center at dig tangent to

Enter by Radius or Diameter

This command will read either "Enter by Radius" or "Enter by Diameter" which allows you to choose whether you enter data in diameters or radii.

Choosing "Enter by Radius" mode causes all subsequent commands which require a radius or diameter to request a radius. The command displayed under the "Arcs "menu will now read "Enter by Diameter".

Choosing "Enter by Diameter" mode causes all subsequent commands which require a radius or diameter to request a Diameter. The command displayed under the "Arcs "menu will now read "Enter by Radius".

Enter Clockwise or CCLW

This command will read either "Enter Clockwise" or "Enter CCLW", which allows you to choose whether you create arcs in a clockwise or counterclockwise (CCLW) direction.

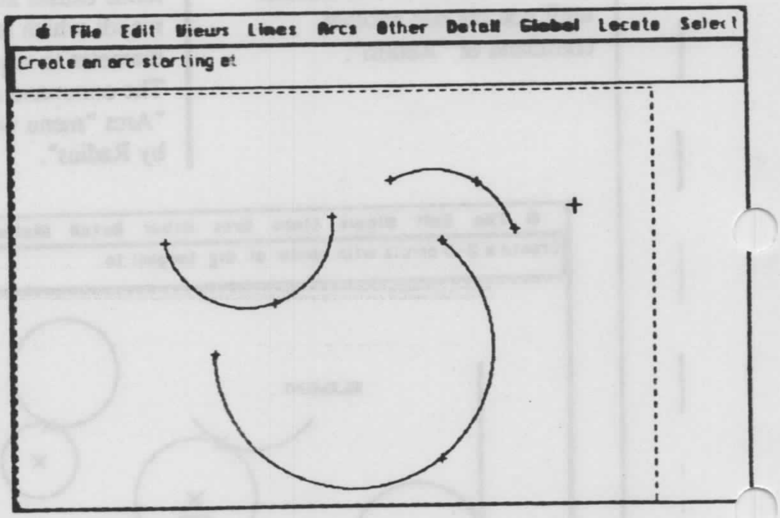
When you choose "Enter Clockwise" mode this causes all subsequent commands which use the arc direction setting to create arcs in the clockwise direction. The command displayed under the "Arcs "menu will now read "Enter CCLW".

When you choose "Enter CCLW" mode this causes all subsequent commands which use the arc direction setting to create arcs in the counterclockwise direction. The command displayed under the "Arcs "menu will now read "Enter Clockwise".

Arc Thru 3 Points

This command allows you to create an arc passing through 3 points.

The arc will be created starting at the first selected point, ending at the third selected point and in the indicated direction. Depending on the direction mode the arc may or may not be generated through the second point, but the second point will lie on the path of the radius describing that arc. If the arc does not pass through the second point, and this is desired, change the direction mode and reselect the points.


File Edit Views Lines Arcs Other Detail Global Locate Select

Create an arc starting at

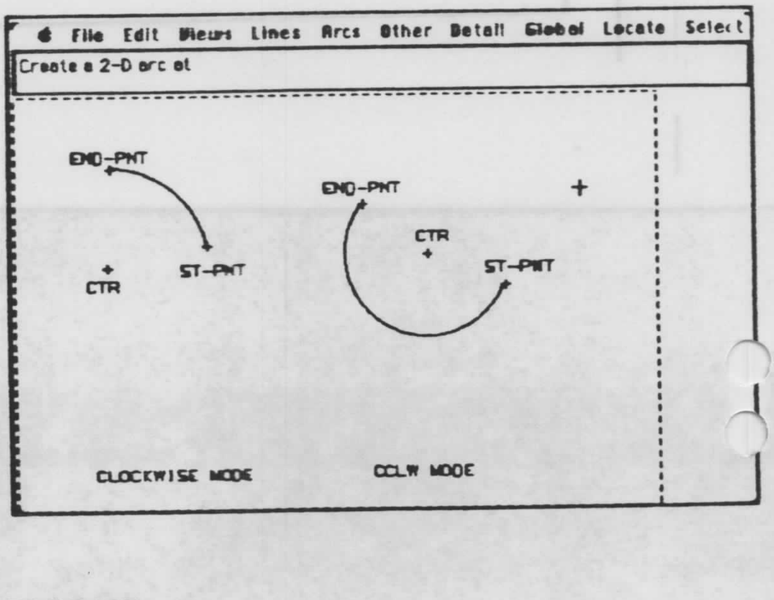
Arc Ctr-SP-EP

This command allows you to create an arc by:

1. **CENTER** - Select a point to define the center.

2. **SP** - Select a starting point for the arc, the radius is then set as the distance between the starting point and the center.

3. **EP** - Select a point near where the arc is to end. The ending point will be set at the intersection of the arc described and a line from the arc center to the point location just described. The direction of the arc is determined by the arc direction mode.


File Edit Views Lines Arcs Other Detail Global Locate Select
END-PNT
ST-PNT
CTR
END-PNT
CTR
ST-PNT
CLOCKWISE MODE
CCLW MODE

Create a 2-D arc at

Arc SA-IA-R-D

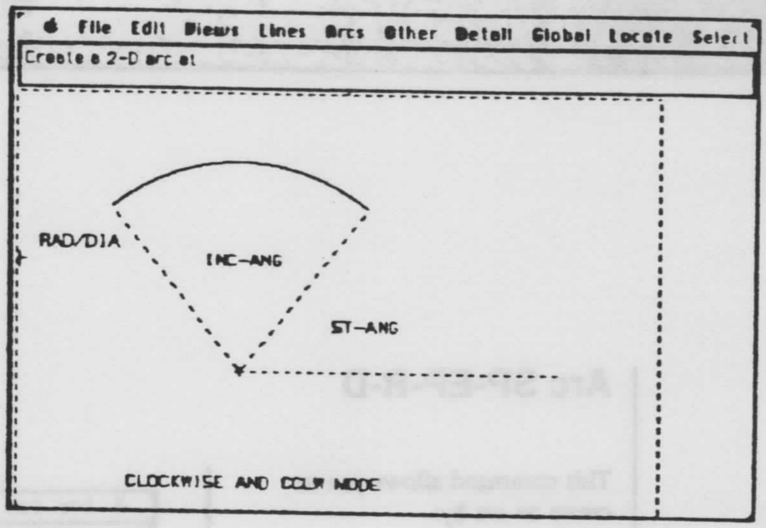
This command allows you to create an arc by:

1. **CENTER** - Select a point to define the center.

2. **R (D)** - Enter a radius or diameter.

3. **SA** - Enter starting angle of the arc (see "Angle Entry" menu in Chapter 1).

4. **IA** - Enter the incremental angle of the arc (in degrees). If this is a positive number, then the direction is the same as the arc direction mode. If negative, then the direction is opposite from the arc direction mode.


File Edit Views Lines Arcs Other Detail Global Locate Select
RAD/DIA
INC-ANG
ST-ANG
CLOCKWISE AND CCLW MODE

Create a 2-D arc at

Arc SA-EA-R-D

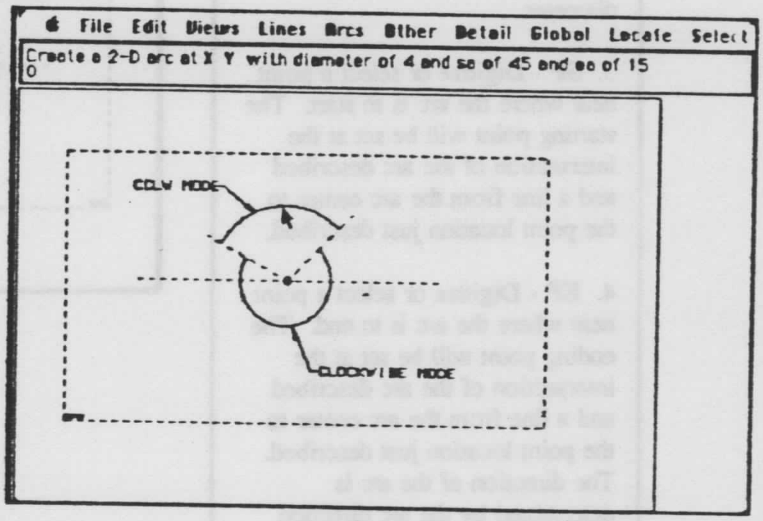
This command allows you to create an arc by:

1. **CENTER** - Select a point to define the center.

2. **R (D)** - Enter a radius or diameter.

3. **SA** - Enter starting angle of the arc (see "Angle Entry" menu in Chapter 1).

4. **EA** - Enter the ending angle of the arc. The direction the arc is drawn is determined by the arc direction setting.


File Edit Views Lines Arcs Other Detail Global Locate Select
CCLW MODE
CLOCKWISE MODE

Create a 2-D arc at X Y with diameter of 4 and sa of 45 and ea of 15
0

Arc SP-EP-R-D

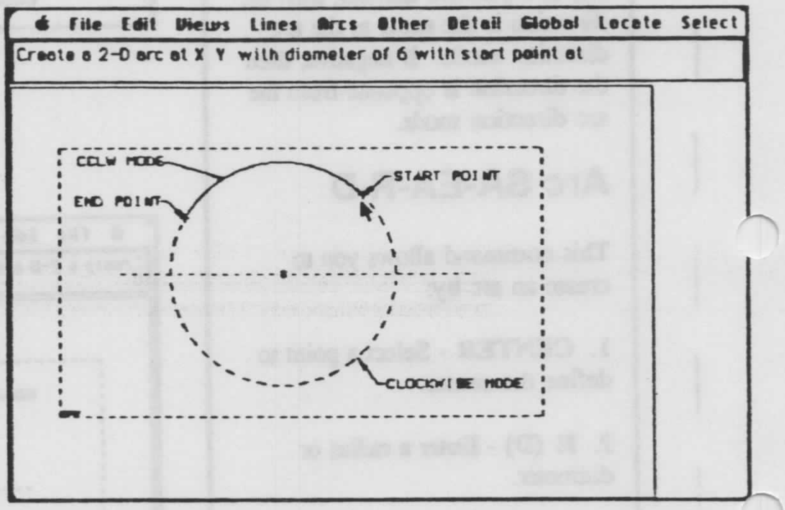
This command allows you to create an arc by:

1. **CENTER** - Select a point to define the center.

2. **R (D)** - Enter a radius or diameter.

3. **SP** - Digitize or select a point near where the arc is to start. The starting point will be set at the intersection of the arc described and a line from the arc center to the point location just described.

4. **EP** - Digitize or select a point near where the arc is to end. The ending point will be set at the intersection of the arc described and a line from the arc center to the point location just described. The direction of the arc is determined by the arc direction mode.



Create a 2-D arc at X Y with diameter of 6 with start point at

Arc SP-IA-R-D

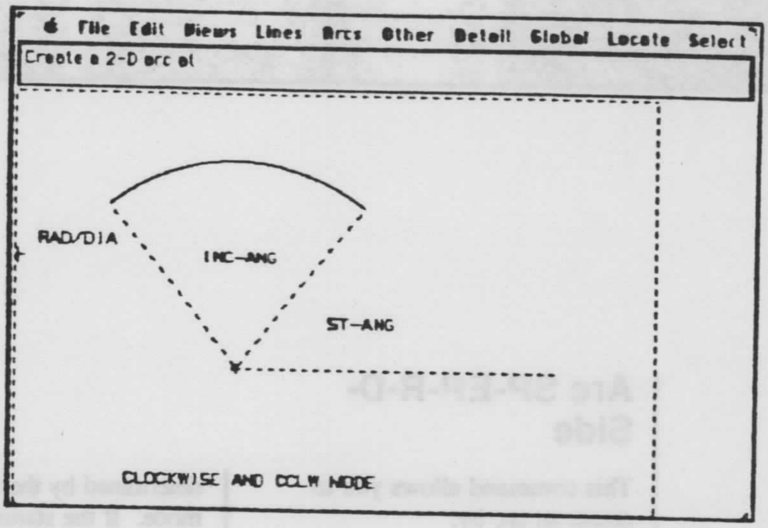
This command allows you to create an arc by:

1. **CENTER** - Select a point to define the center.

2. **R (D)** - Enter a radius or diameter, depending on the selected mode.

3. **SP** - Digitize or select a point near where the arc is to start. The starting point will be set at the intersection of the arc described and a line from the arc center to the point location just described.

4. **IA** - Enter the incremental angle of the arc (in degrees). If this is a positive number, then the direction is the same as the arc direction mode. If negative, then the direction is opposite from the arc direction mode.



```
Create a 2-D arc at
```

Arc SP-EA-R-D

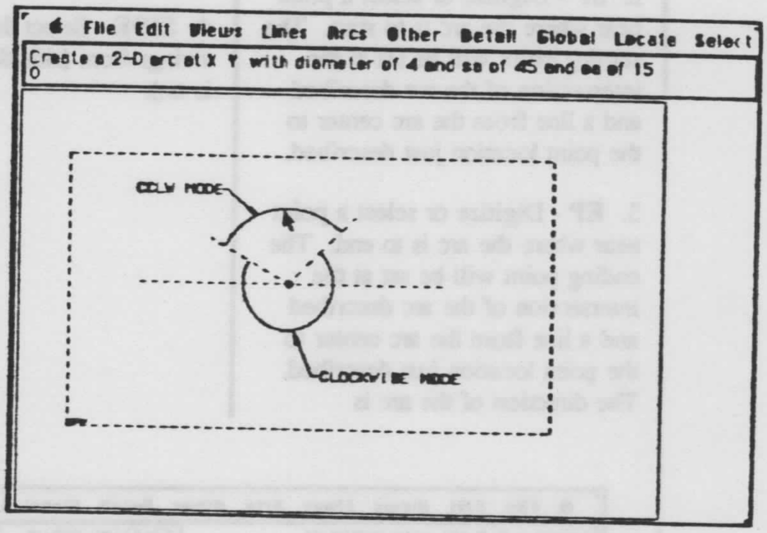
This command allows you to create an arc by:

1. **CENTER** - Select a point to define the center.

2. **R (D)** - Enter a radius of diameter, depending on the selected mode.

3. **SP** - Digitize or select a point near where the arc is to start. The starting point will be set at the intersection of the arc described and a line from the arc center to the point location just described.

4. **EA** - Enter the ending angle of the arc. The direction the arc is drawn is determined by the arc direction setting.



```
Create a 2-D arc at X Y with diameter of 4 and sa of 45 and ea of 15
0
```

Arc SP-EP-R-D-Side

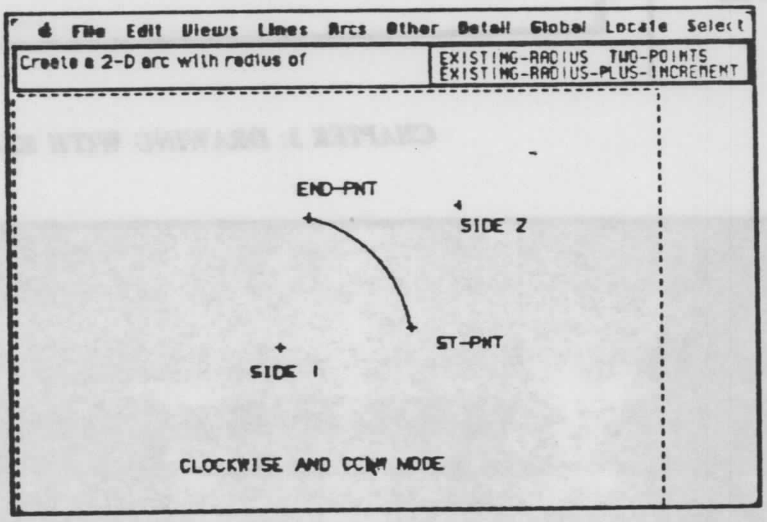
This command allows you to create an arc by:

1. **R (D)** - Enter a radius or diameter.

2. **SP** - Digitize or select a point near where the arc is to start. The starting point will be set at the intersection of the arc described and a line from the arc center to the point location just described.

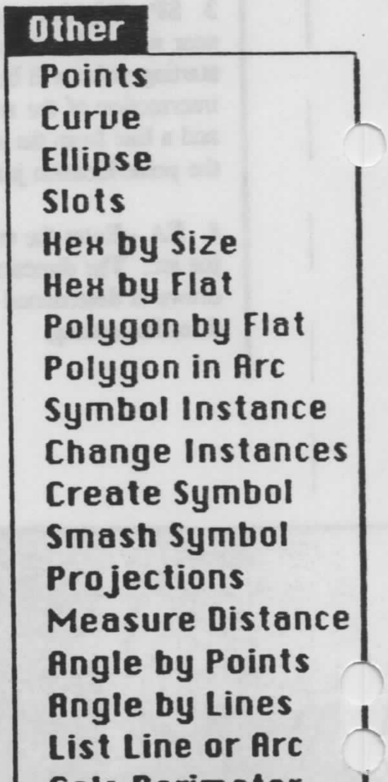
3. **EP** - Digitize or select a point near where the arc is to end. The ending point will be set at the intersection of the arc described and a line from the arc center to the point location just described. The direction of the arc is determined by the arc direction mode. If the starting and ending points you select are farther apart than the entered radius/diameter, selection points for the side of the arc center will not be visible.

4. **SIDE** - Select the side the arc swings from (side the center point is on).


File Edit Views Lines Arcs Other Detail Global Locate Select
EXISTING-RADIUS TWO-POINTS
EXISTING-RADIUS-PLUS-INCREMENT
END-PNT
SIDE 2
ST-PNT
SIDE 1
CLOCKWISE AND CCW MODE

Create a 2-D arc with radius of

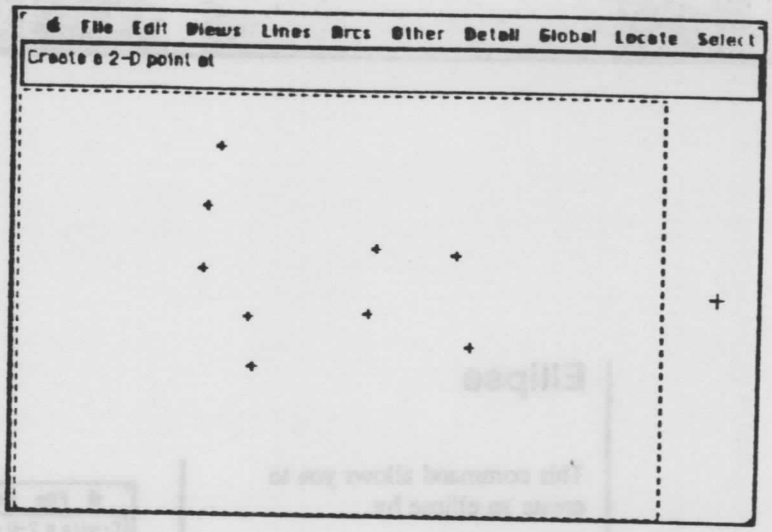
The 18 commands in the "Other" menu allows you to create fitted curves (B-splines), symbols, polygons, closed curves and to measure already existing geometry. The "Projections" command enables you to project geometrey already created in one view to another view using the relationships between view angles.



Points

This command allows you to create points in the current view. A point is created at the point location that you enter using any of the point entry methods (see Chapter 1). Point elements in a symbol can also be selected using the point selection method. At this time, there is only one Point-Font, which is "+".

You can continue to create points in this manner until you choose another command or "Return".



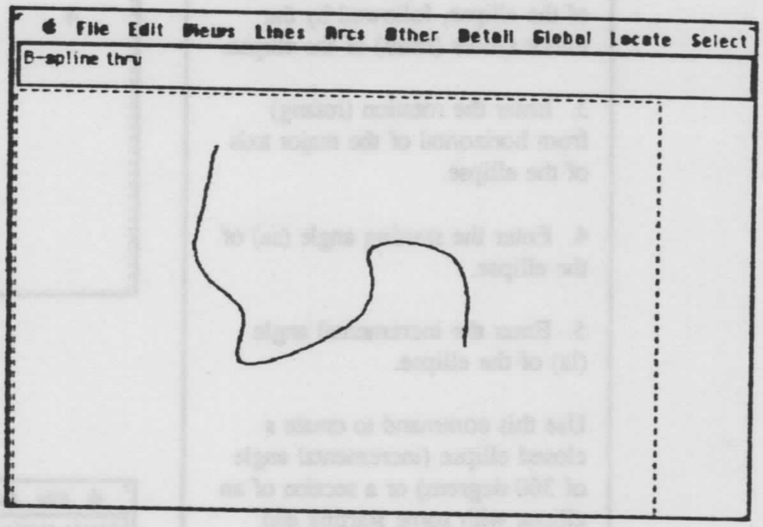
Create a 2-D point at

Curves

This command allows you to create a smooth fitting curve through all of the points that you have entered. To create a B-Spline curve:

1. Enter the points in the order which they are to be passed through. You need to have a minimum number of four (4) points to create the B-Spline (a smooth curve through three points would give you a circle not a B-Spline , therefore nothing is created).

2. When you have finished entering points, choose "Enter" and a smooth fitting curve will be created. After the curve has drawn you will be returned to the Menu Bar.



B-spline thru

Ellipse

This command allows you to create an ellipse by.

1. Enter the point location of the center of the ellipse.

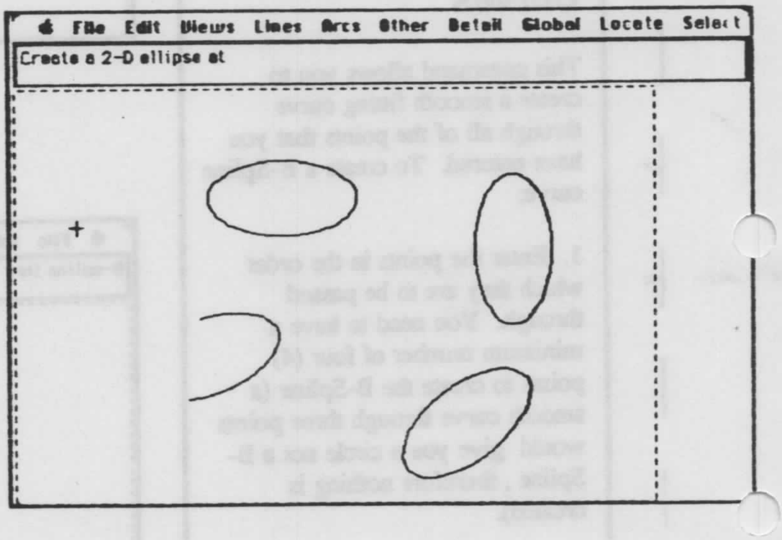
2. Enter the major radius (rmaj) of the ellipse, followed by the minor radius (rmin) of the ellipse.

3. Enter the rotation (rotang) from horizontal of the major axis of the ellipse.

4. Enter the starting angle (sa) of the ellipse.

5. Enter the incremental angle (ia) of the ellipse.

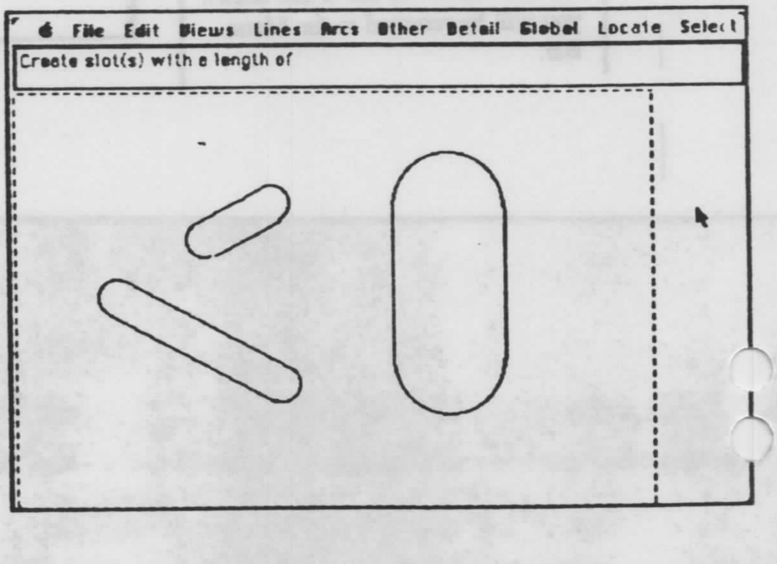
Use this command to create a closed ellipse (incremental angle of 360 degrees) or a section of an ellipse with some starting and incremental angle.



Create a 2-D ellipse at

Slots

This command allows you to create a slot of a specified width, length, and angle at a given location. A slot consists of two parallel lines and two semi-circles at opposing ends. Once created, the lines and arcs can be selected as individual elements. To create a slot:



Create slot(s) with a length of

1. Enter the length of the slot that you want to create. This length includes the arcs at the ends.

2. Enter the width of the slot (this width is also the diameter of the arcs).

3. Enter the angle of the slot (an angle of 0 degrees is horizontal).

4. Select the point location for the center of the slot.

You can continue to enter this slot at different center point locations, until you choose "Return". If you choose "Return" at any time while entering information, the "Ellipse" command will be terminated and you will be returned to the Menu Bar.

Hex by Size

This command allows you to create a Hex-head (6-sided polygon) of a specific size, centered at a point that you select.

1. Enter the size (numeric value) of the Hex-head. This size represents the distance between two opposite flats. A flat is one of the six sides.

2. Enter the center location of the Hex-head. You can continue to enter this Hex-head at different center point locations, until you choose "Return".

This command also allows you to create bolt heads by size as well as allen heads.

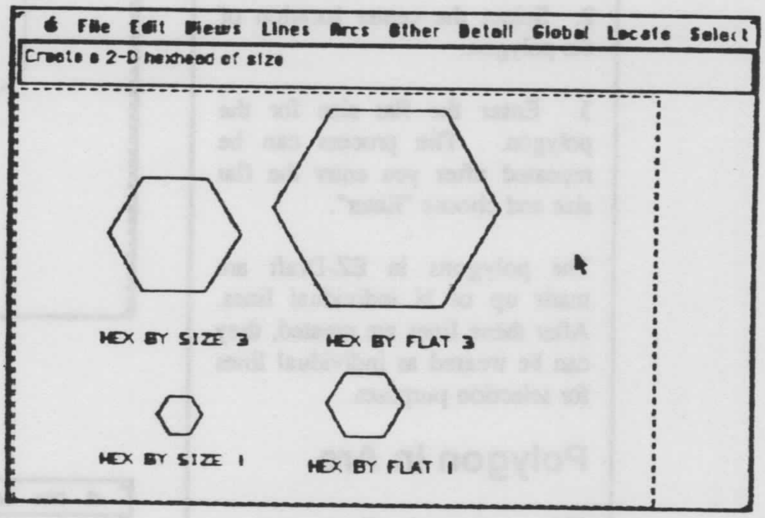
Hex by Flat

This command allows you to create a Hex-head (6-sided polygon) with the given length per flat and centered at the entered point.

1. Enter a value to be used as the flat length per side.

2. Enter the center location of the Hex-head. You can continue to enter this Hex-head at different center point locations, until you choose "Return".

The Hex-head is actually six separate lines, and each line can be treated individually.


File Edit Views Lines Arcs Other Detail Global Locate Select
HEX BY SIZE 3
HEX BY FLAT 3
HEX BY SIZE 1
HEX BY FLAT 1

Create a 2-D hexhead of size

Polygon by Flat

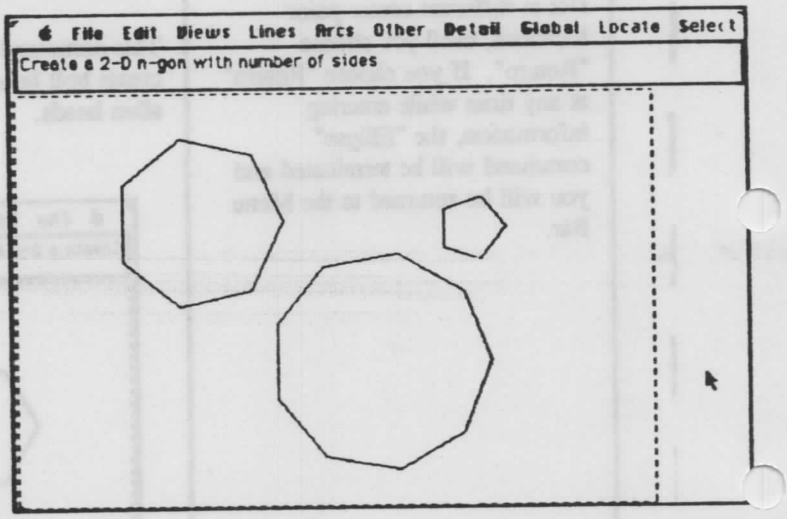
This command allows you to create a polygon of N sides, with a specific flat size centered at the entered location.

1. Enter the number of sides (N).

2. Select the center location of the polygon.

3. Enter the flat size for the polygon. The process can be repeated after you enter the flat size and choose "Enter".

The polygons in EZ-Draft are made up of N individual lines. After these lines are created, they can be treated as individual lines for selection purposes.


File Edit Views Lines Arcs Other Detail Global Locate Select

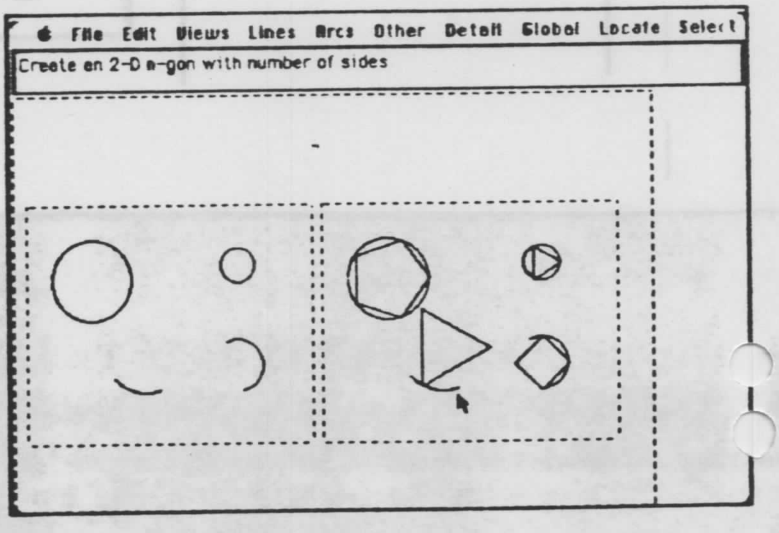
Create a 2-D n-gon with number of sides

Polygon in Arc

This command allows you to create an N-sided polygon inscribed in an arc that you select.

1. Enter the number of sides (N) that the polygon will have.

2. Select the Arc in which the polygon will be inscribed.


File Edit Views Lines Arcs Other Detail Global Locate Select

Create an 2-D n-gon with number of sides

Symbols

The next four commands, "Symbol Instances", "Change Instances", "Create Symbol" and "Smash Symbol", in the "Other" menu comprise the symbol capability of the EZ-Draft system. A symbol is a group or "collection" of previously defined geometry that you use more than once while creating a drawing(s) on the EZ-Draft system.

Currently, a symbol consists of points, lines, arcs, and text. To define a symbol, you must select geometry that will make up that symbol, then indicate the symbol origin. Once a symbol has been created, it can be used to create symbol instances.

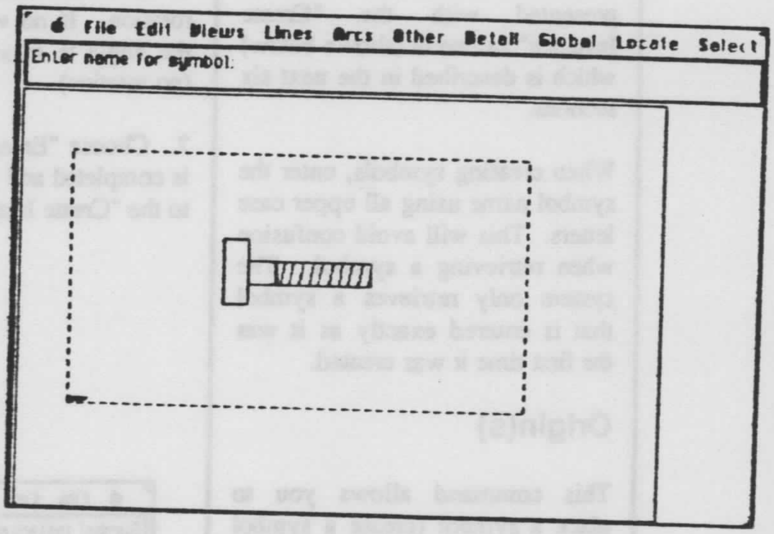
A symbol instance is an occurrence of a symbol. Each symbol instance has its own origin (which is where the symbol origin is placed), scale factors (x and Y), angle of rotation, and mirroring capability.

Since points in a symbol can be referenced with the existing point option, points should be placed at locations in a symbol where a line could connect, or where text could be placed. For example, in an electrical symbol, points could be placed to connect wiring lines, or they could be placed in a drawing format symbol for locating the text.

Once a symbol instance is created, it can only be selected as a whole by its origin. However, point elements in a symbol may be referenced by using the "Select" menu. No other elements in a symbol can be referenced.

If the geometry in a symbol instance (other than points) is needed for selection or modification, the symbol instance can be smashed (broken down into points, lines, arcs, and text). This geometry will then be exactly like all other points, lines, arcs, and text.

It is possible to change all occurrences of a symbol in a part. (This might be desirable if all 1/4" pins were being changed to 5/16" pins). This is done with the "Change Instances" command.



Enter name for symbol:

Symbol Instance

This selection initiates the creation of symbol instances. Enter the alpha-numeric name of the symbol to be used. If the name is valid (a symbol of that name exists in the part and has been saved), then you will be presented with the "Create Instance" submenu (shown below) which is described in the next six sections.

When creating symbols, enter the symbol name using all upper case letters. This will avoid confusion when retrieving a symbol. The system only retrieves a symbol that is entered exactly as it was the first time it was created.

Origin(s)

This command allows you to place a symbol (create a symbol instance) where you want using the current Angle, Scaling, and Mirroring at selected point location. This location becomes the symbol instance origin. Then the Mirroring, if any, is applied. Next, the Scaling (if not 1:1) is applied. Finally, the Angle of rotation (if not 0) is applied.

You may continue to create this "Symbol Instance" by selecting new point locations for the symbol origin.

Set-Angle

This command allows you to set the angle for future "Symbol Instances".

1. Enter the value of the angle of rotation. If no value is entered, the angle is assumed to be zero (no rotation).

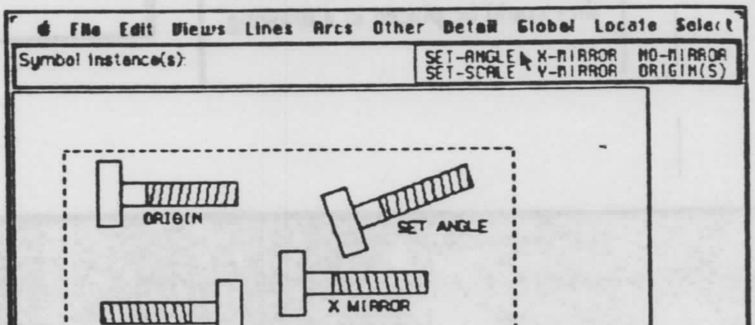
2. Choose "Enter" the command is completed and you are returned to the "Create Instance" submenu.

Set-Scale

This command allows you to set the scale for future symbol instances.

1. Enter the scale factor for the symbol.

2. Choose "Enter" the command is completed and you are returned to the "Create Instance" submenu.



Symbol Instance(s):

X-Mirror

Choosing this command will apply Mirroring about the X axis for future symbol instances. The Mirroring is applied to the symbol before Scaling and Rotating. You will remain in the "Create-Instance" submenu.

Y-Mirror

Choosing this command will apply Mirroring about the Y axis for future symbol instances. The Mirroring is applied to the symbol before Scaling and Rotating. You will remain in the "Create-Instance" submenu.

No-Mirror

This command allows you to turn Off the Mirroring for future symbol instances. When you select this command, all current Mirroring is also turned Off. You will remain in the "Create-Instance" submenu.

Change Instances

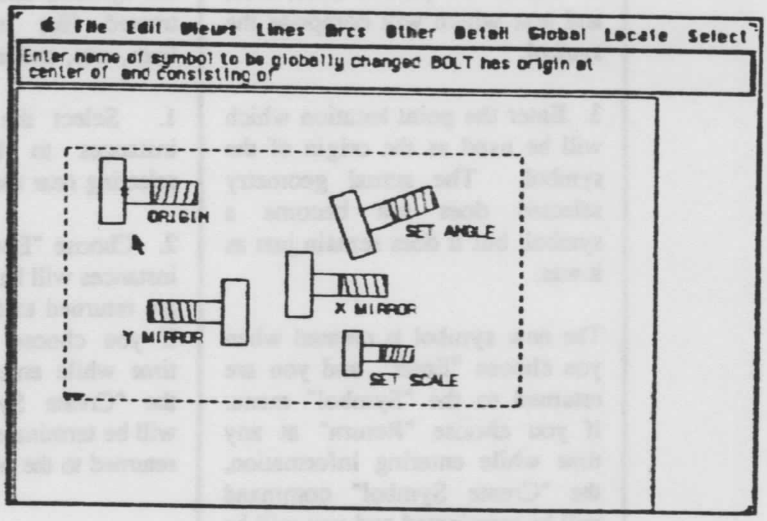
This command allows you to change all instances of a symbol.

1. Enter the alpha-numeric name of an existing symbol.

2. Select the points, lines, arcs, and text which will comprise the new symbol.

3. Enter the point location that will be used as the origin of the new symbol. The actual geometry selected does not become a symbol, the selected geometry will be stored as a symbol.

All symbol instances of the symbol name are automatically changed to the new symbol. Any Angle, Scaling or Mirroring is still applied. The new symbol is created when you choose "Enter", and you are returned to the "Symbol" menu. If you choose "Return" at any time while entering information, the "Change Instances" command will be terminated and you will be returned to the Menu Bar.



Enter name of symbol to be globally changed BOLT has origin at center of and consisting of

Create Symbol

This command allows you to create a symbol.

1. Enter the alpha-numeric name to be used for the symbol.

2. Select the points, lines, arcs, and text which will comprise the symbol.

3. Enter the point location which will be used as the origin of the symbol. The actual geometry selected does not become a symbol, but it does remain just as it was.

The new symbol is created when you choose "Enter", and you are returned to the "Symbol" menu. If you choose "Return" at any time while entering information, the "Create Symbol" command will be terminated and you will be returned to the Menu Bar.

Smash Symbol

This command allows you to smash existing symbol instances into their component parts (points, lines, arcs, and text). Once a symbol instance is smashed, it loses its connection to the symbol, and the geometry is treated like individual points, lines, arcs, and text.

1. Select the desired symbol instances to be smashed by selecting near the origin.

2. Choose "Enter" The symbol instances will be smashed and you are returned to the symbol menu. If you choose "Return" at any time while entering information, the "Create Symbol" command will be terminated and you will be returned to the Menu Bar.

Projections

"Projections" contains commands needed for projection.

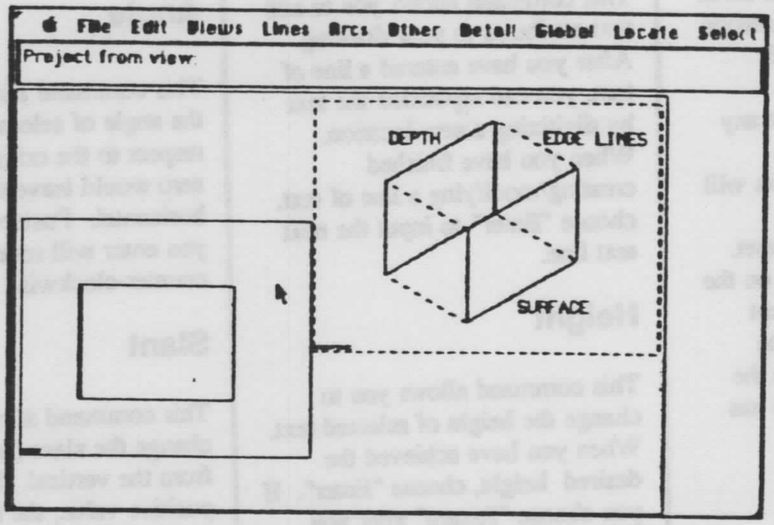
When you choose "Projection" from the "Other" menu, the following prompt will appear: "Project From View:". With the pointer, select the view containi the elements that you wish to project. A submenu will appear (see the figure bottom of next page).

When working with the commands in the "Projection" menu, select elements for projection carefully, in order avoid lines being drawn on top of other lines. You must set both the "Surface" and "Depth" values before projecting elements into another view.

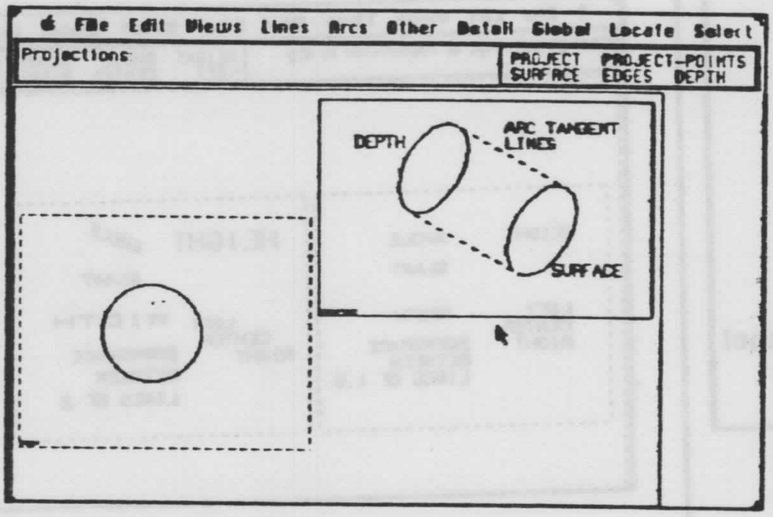
Surface

When you select this command, the following prompt will appear: "Project from a depth of". Enter the desired value, then choose "Enter". After you have entered the surface height value, the "Change Settings" submenu (see Chapter 1) will appear.

You can change any or all of these settings, if you choose not change any of these setting, the current system settings will be used. After you have changed the


File Edit Views Lines Arcs Other Detail Global Locate Select
DEPTH
EDGE LINES
SURFACE

Project from view:



Projections:

desired settings, choose "Enter" and you will be returned to the "Projection" submenu.

Depth

When you select this command, the following prompt will appear: "Project to a depth of". Enter the desired depth value, then choose "Enter". After you have entered the depth value, the "Change Settings" submenu will appear.

You can change any or all of these settings, if you choose not to change any of these settings, the current system settings will be used. After you have changed the desired settings, choose "Enter" and you will be returned to the "Projection" submenu.

Edges

This setting is optional. Edges allows you to change the characteristics of the edge lines in the projected view.

Project-Points

This command allows you to project selected edge lines into one or more views, at points that you select.

Measurement

These last 5 commands in the "Other" menu allow you to measure aspects of geometry that has already been created.

Measure Distance

This command allows you to measure or verify information about a specific element, between selected points.

Angle by Points

This command allows you to measure an angle determined by three points. Enter the three points using any of the point entry methods (see Chapter 1). The system will calculate the included angle (less than 180 degrees) by assuming an imaginary line between the first and second point, and between the second and third points. The angle will then be displayed in the prompt area of the screen.

Choose "Return" when you are finished viewing the information, and you will be returned to the Menu Bar. Selecting another command will also exit you from this command.

Angle by Lines

This command allows you to measure the angle between two selected lines. After selecting the two lines, the sytem will then display the angle measurement in the prompt area of the screen.

Choose "Return" when you are finished viewing the information, and you will be returned to the Menu Bar. Selecting another command will also exit you from this command.

List Line or Arc

This command allows you to display specific data about a selected element. Typical information would be view, color, layer, font, start point, and endpoint. The information displayed will vary depending on the type of element you choose.

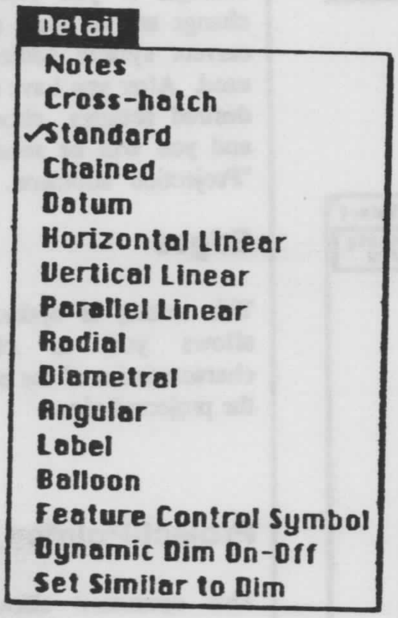
Choose "Return" when you are finished viewing the information, and you will be returned to the Menu Bar. Selecting another command will also exit you from this command.

Calculate Perimeter

This command calculates the distance traveled along a selected perimeter (boundary). The perimeter may be selected by using any of the multiple selections methods, and does no have to be a continuous perimeter. After your selection has been made, the system will automatically display the perimeter distance in the prompt area of the screen.

Choose "Return" when you are finished viewing the information, and you will be returned to the Menu Bar. Selecting another command will also exit you from this command.

The "Detail" menu contains all of the commands used to dimension and annotate your EZ-Draft drawings. If you set new dimensioning parameters at any time while working with a drawing, the new parameters will be used for all subsequent dimensions until they are reset. All dimensions created are on the current Layer, use the current Color, are displayed with the current Curve-font, and use the current Group-name (if one has been set).



Notes

This command allows you to add text attributes to your drawing. After you have entered a line of text, you can reposition the text by digitizing a new location. When you have finished creating/modifying a line of text, choose "Enter" to input the next text line.

Height

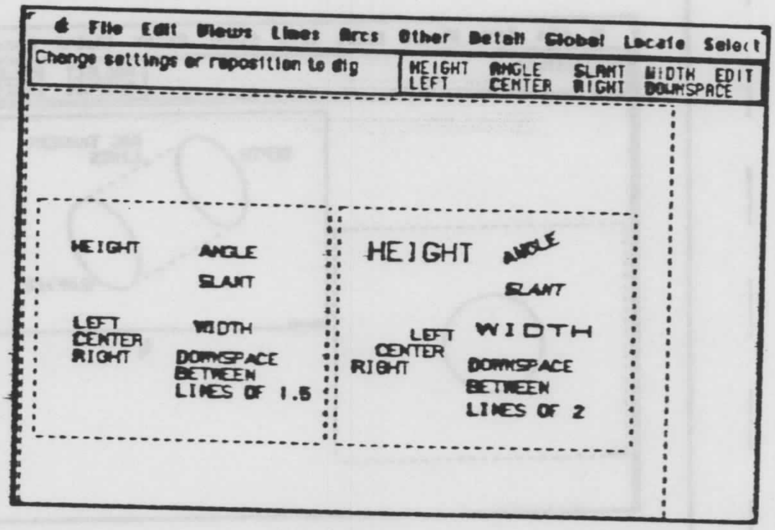
This command allows you to change the height of selected text. When you have achieved the desired height, choose "Enter". If you choose "Return" after you select this command, the height will remain unchanged.

Angle

This command allows you to set the angle of selected text with respect to the origin. A value of zero would leave the text horizontal. Positive values that you enter will rotate the text counter-clockwise.

Slant

This command allows you to change the slant (angle) of the text from the vertical. If you enter a positive value, the text will slant to the right. If you enter a negative value, the text will slant to the left. All values should be entered in degrees.


Change settings or reposition to dig

Width

This command allows you to change the width to height ratio of the text. For example, a ratio of .5 would make the text half as wide as it is tall. The default setting ratio is 1:1.

Left Center Right (justification)

These commands allow you to place the text with respect to the origin that you have digitized. For instance, "Left" justification places the extreme left text character at the origin. "Right" justification places the extreme right text character at the origin. "Center" justification places the center of the text at the origin.

Downspace

This command allows you to set the value of the space between multiple lines of text. A value of 1 is for no space, a value of 1.5 is for one half of the text height space, and a value of 2 is for 1 space of text height. Any value may be entered with this command.

Edit

After choosing this command you will be prompted to select the text for editing. The selected text will appear in the prompt area of the screen. This is a single line editor so only one line of text can be edited at a time. You may select any or all of the text that appears in the prompt area much the same as in other Macintosh applications.

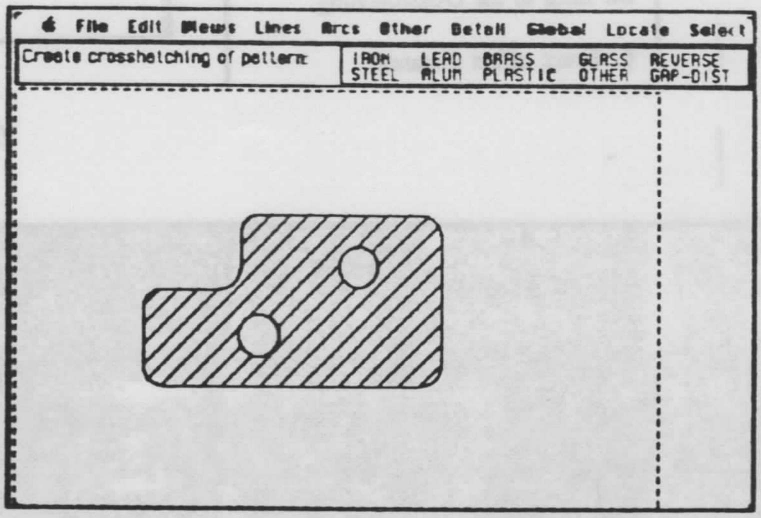
Crosshatch

With the "Crosshatch" command, you can generate a variety of crosshatching patterns.

1. Choose a crosshatching pattern to be created (see appendix under crosshatching).

2. With the cursor, select the entities that form a closed chain for the outer boundary and optional internal holes. This is the area that will be crosshatched.

3. Choose "Enter" to complete the command.



Create crosshatching of pattern:

Reverse

This command allows you to reverse the angle of the crosshatch.

1. Choose "Reverse".

2. Choose the desired crosshatching pattern.

3. With the pointer, indicate the area for the reverse crosshatch.

4. Choose "Enter" to complete the command.

Gap-Dist

This command allows you to control the basic distance between repeating crosshatch patterns.

Other

This command allows you to create your own crosshatching pattern.

1. Choose "Other".

2. Enter the angle, curve-font and color you wish it to be drawn.

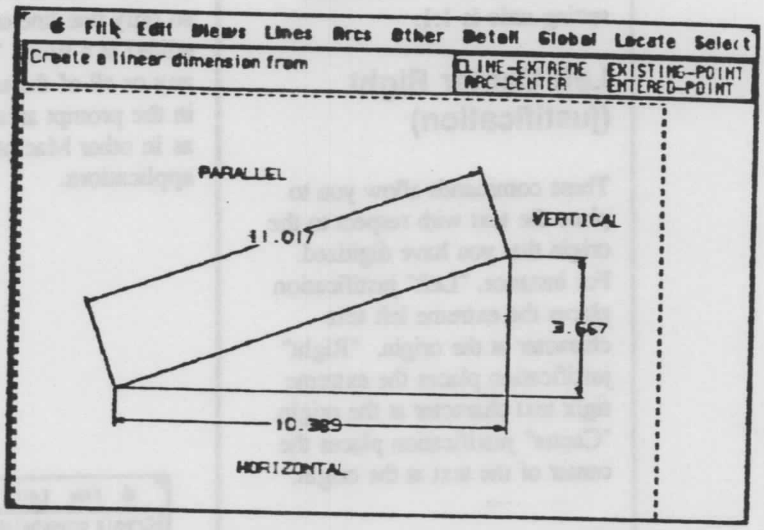
5. Enter spacing distance between the lines in the crosshatching.

6. Enter offset distance.

There are three ways of describing a linear dimension in the EZ-Draft system "Horizontal Linear", "Vertical Linear" and "Parallel Linear". These dimensions can be represented in three different modes "Standard", "Chained" or "Datum". EZ-Draft allows you to tailor your dimensioning methods to your own particular needs.

Standard

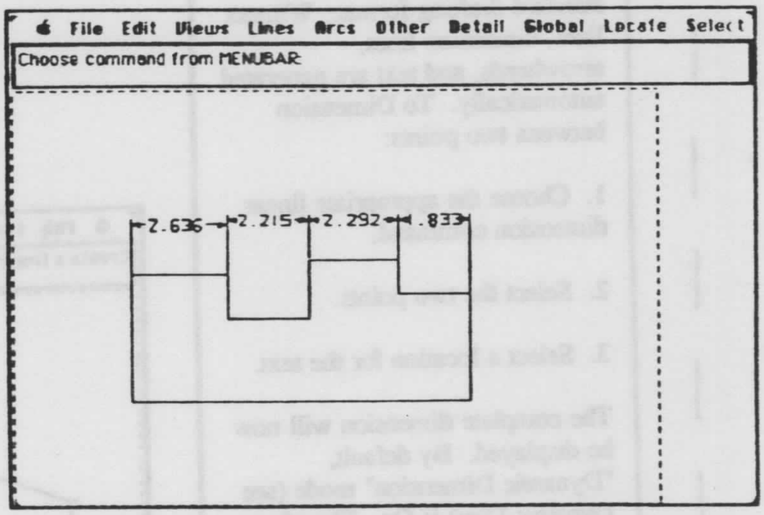
This dimensioning mode allows you to define each linear dimension by entering the two points to be dimensioned. To do this, select the appropriate dimension command "Horizontal Linear", "Vertical Linear", or "Parallel Linear". Choose "Enter" to complete the command.



Create a linear dimension from

Chained

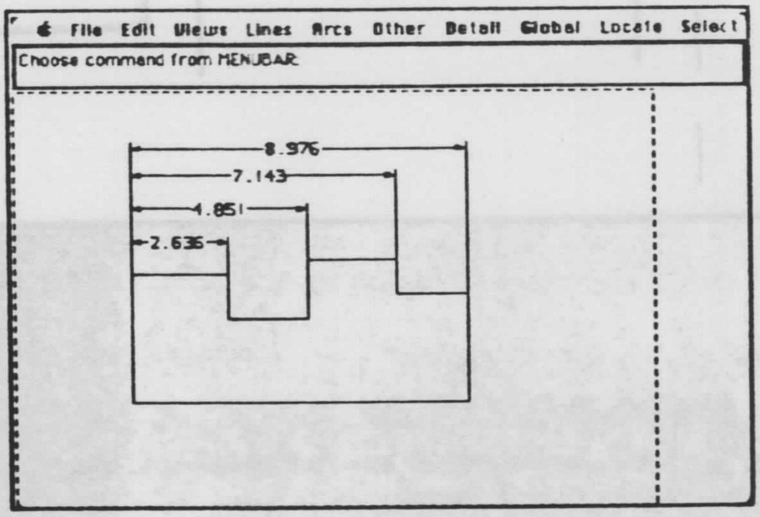
This dimensioning mode allows you to dimension a chain of linear dimensions (See Figure). For example, after the initial linear dimension, the first point of the next dimension is assumed to be the last point of the previous dimension. Choose "Enter" to complete the command.


File Edit Views Lines Arcs Other Detail Global Locate Select
2.636
2.215
2.292
.833
Choose command from MENUBAR

Datum

This dimensioning mode allows you to define dimensions by indicating only the second point (the first point is the datum or baseline). Choose "Enter" to complete the command.

The "Datum" mode includes automatic witness line suppression of all except the longest datum witness line.


File Edit Views Lines Arcs Other Detail Global Locate Select
8.976
7.143
4.851
2.636
Choose command from MENUBAR

Horizontal Linear
Vertical Linear
Parallel Linear

These commands allow you to measure the distance between two points. The "Horizontal Linear" command will display the horizontal distance between the two point selections. The "Vertical Linear" command will display the vertical distance between the two point selections. The "Parallel Linear" command will display the parallel distance between the two point selections. These distances are extracted from the geometry and presented in standard drafting format. Witness lines, dimension lines, arrowheads, and text are generated automatically. To Dimension between two points:

1. Choose the appropriate linear dimension command.

2. Select the two points.

3. Select a location for the text.

The complete dimension will now be displayed. By default, "Dynamic Dimension" mode (see Dynamic Dim) is On. Therefore, after the dimension is defined, you can modify the text location and parameters. To complete the command, choose "Enter".

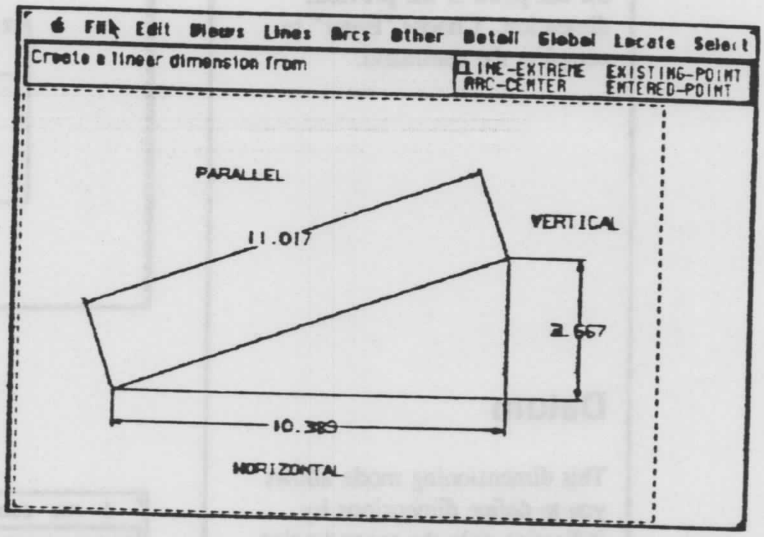
The dimensions you create with these commands have several parameters. The parameters have initial values when the program begins, but they can be changed each time you create a new dimension. If you set new parameters, they will be used for each successive dimension until you change them again.

Linear Dynamic Dimension Submenu

The selections in this submenu allow you select points for dimensioning using four modes of selection.

1. **Existing point** - This mode of selection allows you to create a linear dimension between two existing points. You can continue to use the point selection mode until you select a different mode.

2. **Entered Point** - This mode of selection allows you to create a linear dimension between two points that you enter using any of the point selection methods (see Chapter 1).



Create a linear dimension from

3. **Arc Center** - This mode of selection allows you to create a linear dimension between two selected Arc Centers or an Arc Center and a location using any of the other three modes. You can continue to select Arc Centers until you select a different mode.

4. **Line Extreme** - This mode of selection allows you to create a linear dimension from ends of selected elements. "Line Extreme" is the default mode of selection for dimensions. All selections will be in "Line Extreme" until you choose a different mode.

While using the "Linear Dimension" submenu, this submenu will appear each time you select one of the four modes explained above. This submenu contains commands which allow you to modify the dimensions you have created.

Arrows

Arrows is a toggle switch that allows you to place the arrows either inside or outside of the witness lines. Your selection automatically becomes the default for the next dimension.

Par Hor Ver

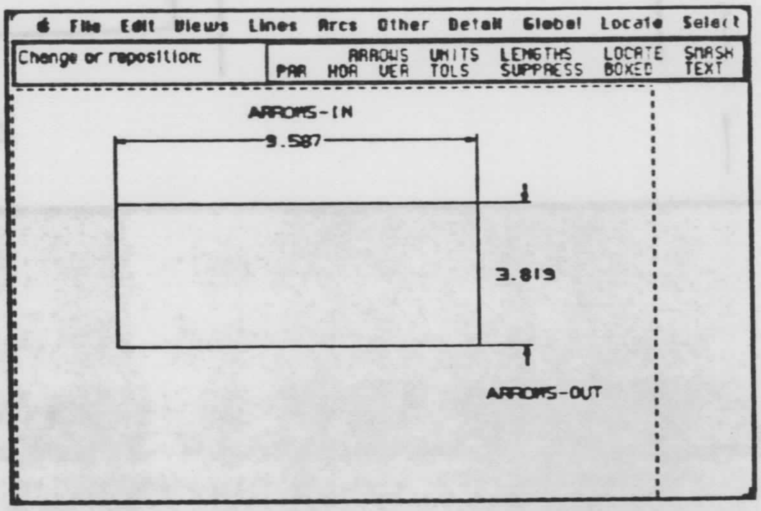
These commands allow you to change the current dimension type for modification after your initial selection of parallel, horizontal or vertical. "Par" redefines the current dimension to be parallel. "Hor" redefines the current dimension to be horizontal. "Ver" redefines the current dimension to be vertical.

Units

Each time you choose "Units", two choices will appear:

1. **Dimension** This command redefines the type of units (metric or English) and the number of decimal places that indicate the numeric values of the basic dimension. The default setting is decimal.

2. **Tolerance** This command redefines the type of units (metric or English) and the number of places that indicate the accuracy of the tolerance values. The default setting is decimal.



Change or reposition:

When you select either command, the submenu below will appear.

Decimal

This command allows you to present the current dimension in whole numbers, with the remainder displayed as a decimal. You can choose the number of decimal places, (e.g. 112.250). This is the default dimensioning presentation display.

Fraction/Base

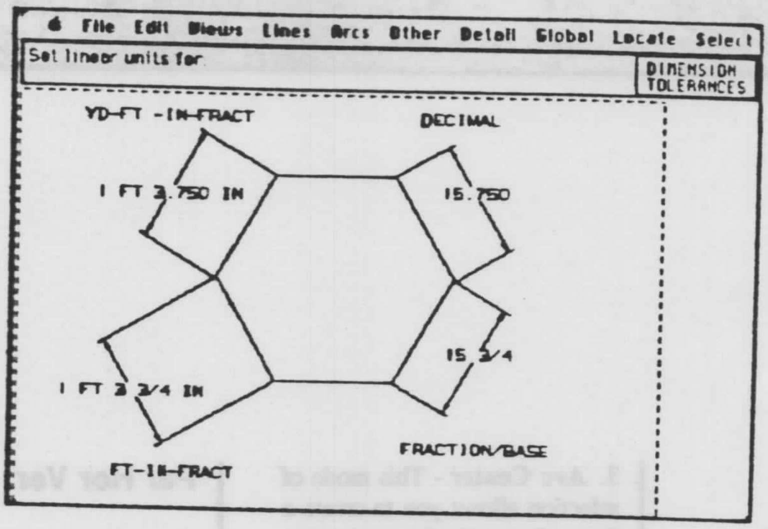
This command allows you to set the minimum denominator for fraction. For example, if you enter a value of 4, then all numbers will be rounded to the nearest 1/4 fractional unit.

Ft-In-Fract

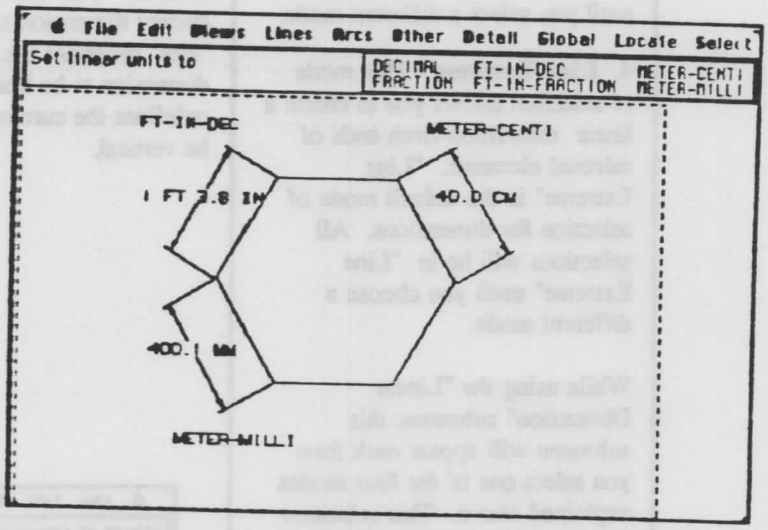
This command allows you to display the current dimension in units of feet, inches and fractions of inches (e.g. 9ft. 4-1/3 in.).

Yd-Ft-In-Fract

This command allows you to display the current dimension in yards, feet, inches, and fractions of inches (e.g. 3 yd. 0 ft. 4-1/4 in.).



Set linear units for



Set linear units to

Ft-In-Dec

This command allows you to display the current dimension in feet, and inches, with the remainder in decimal inches (e.g. 3 ft. 4.25 in).

Meter-Milli

This command allows you to present the current dimension in meters, with the remainder as decimal millimeters (e.g. 2 m 150.5 mm).

Meter-Cent

This command allows you to display the current dimension in meters, with the remainder as decimal centimeters (e.g. 2 m 15.5 cm).

Tols Submenu

The commands in this submenu add tolerance information to the current dimension available for modification. Tolerances include limit dimensions, maximum/minimum increment, and single tolerance variable with a plus and minus sign. Values and display can be modified. The tolerance defined for the current dimension becomes the default for the next dimension, unless you decide to change it.

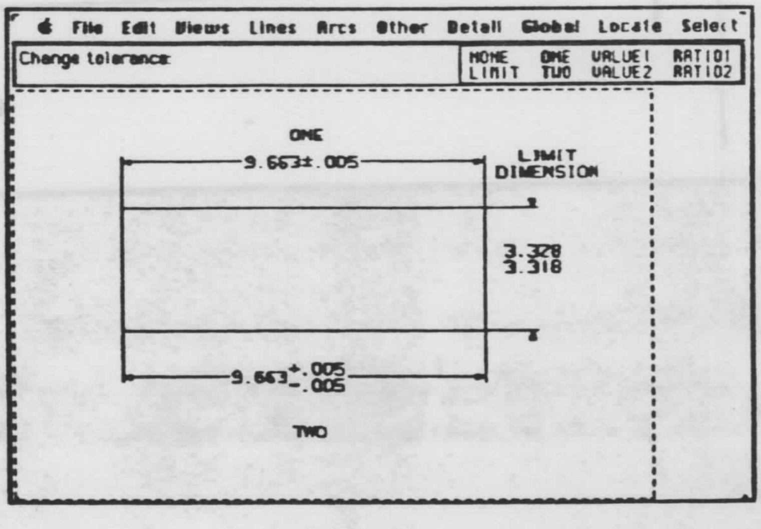
Each time you choose "Tols", the following submenu will appear:

None

This command allows you to remove the tolerance displayed for the current dimension, leaving only the primary value.

One

This command allows you to add a single value (with a plus + and a minus - sign) to the primary value of the current dimension. The tolerance value can be set with the "Value 1" command.



Change tolerance:

Two

This command allows you to add two values (one with a plus + sign and one with a minus - sign) to the primary value of the current dimension . The tolerance values can be set with the "Value 1" and "Value 2" commands.

Limit Dimensions

This command allows you to redisplay the primary dimension value of the current dimension as two numbers, one with a tolerance value added, the other with a tolerance value subtracted. The tolerance values can be set with the "Value 1" and "Value 2" commands.

Value 1

This command allows you to redefine the value of a single tolerance dimension, or the first tolerance of a two-value tolerance. The initial default for inches is .005. The initial default for millimeters is 1.

Value 2

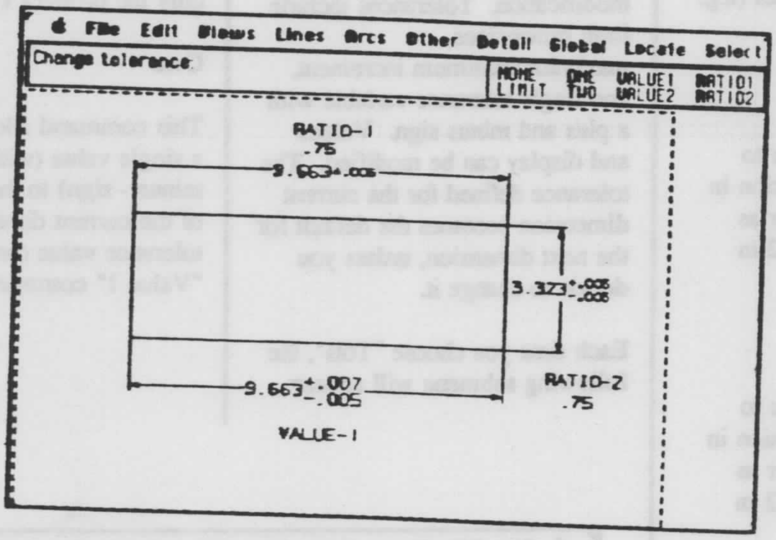
This command allows you to redefine the value of the second tolerance of a two-value tolerance. The initial default for inches is .005. The initial default for millimeters is 1.

Ratio-1

This command allows you to set the height of one tolerance as a percentage of the height of the main text.

Ratio-2

This command allows you to set the height of two tolerances as a percentage of the height of the main text.



Change tolerance:

Lengths Submenu

Each time you choose "lengths", the submenu of commands in the figure on the right will appear.

The commands in this submenu allow you to control dimensioning standards (e.g. length of witness lines, gap size between dimensioning lines).

Broken

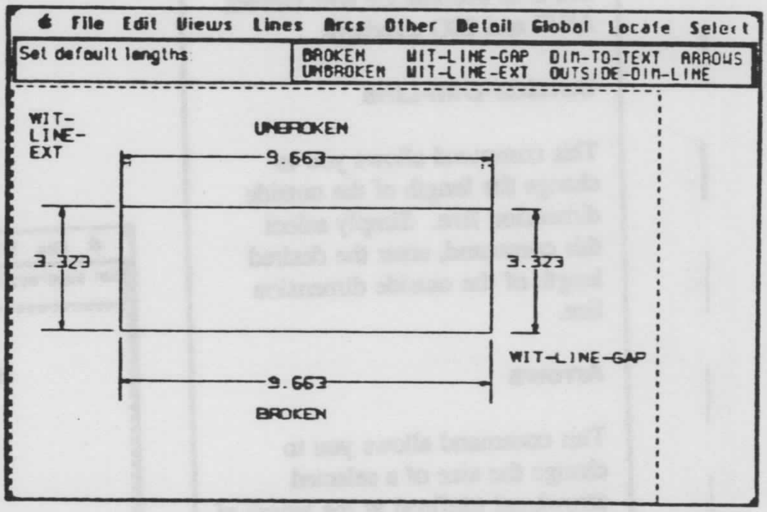
This command allows you to "break" the dimension line and then display the text within the gap that is subsequently created. This is the initial default setting for dimensioning.

Unbroken

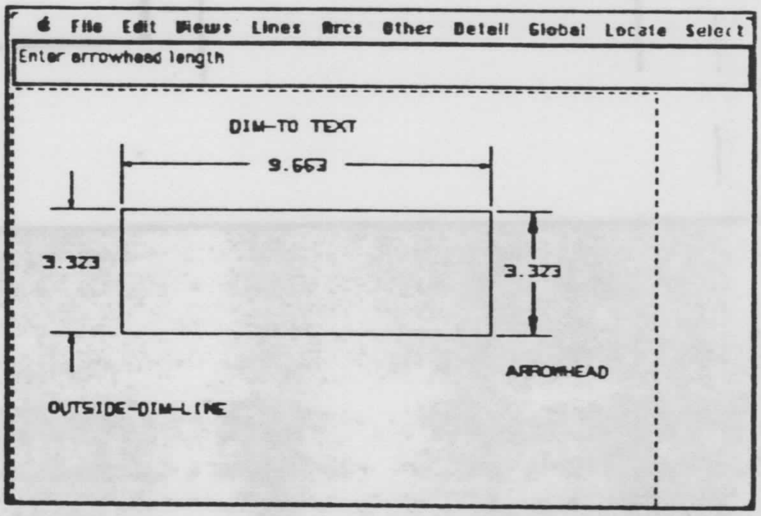
This command allows you to display the current dimension line as a continuous line. When you use this command, text will be displayed above the dimension line.

Wit-Line-Gap

This command allows you to change the length of the gap between the witness line and the geometry that is being dimensioned. The default size is in accordance with current ANSI and ISO standards.



Set default lengths


File Edit Views Lines Arcs Other Detail Global Locate Select
DIM-TO TEXT
9.663
3.323
3.323
ARROWHEAD
OUTSIDE-DIM-LINE

Enter arrowhead length

Wit-Line-Ext

This command allows you to change the distance that the witness line extends beyond (above) the dimension line. The default size is in accordance with current ANSI and ISO standards.

Dim-To-Text

This command allows you to change the distance from the dimension line to the text value of a broken dimension. The default size is in accordance with current ANSI and ISO standards.

Outside-Dim-Line

This command allows you to change the length of the outside dimension line. Simply select this command, enter the desired length of the outside dimension line.

Arrows

This command allows you to change the size of a selected arrowhead (defined as the length of the arrowhead in drawing units). The default size is in accordance with current ANSI and ISO standards.

1. Choose "Arrows".

2. Enter the desired length for the arrowhead.

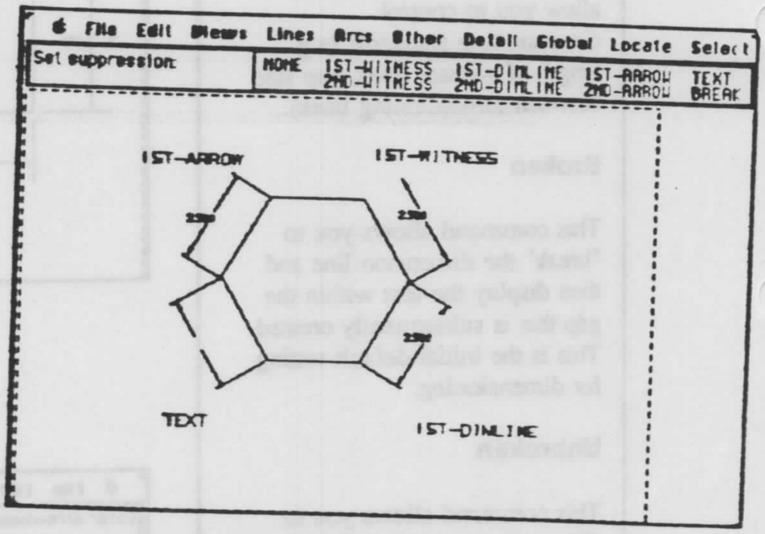
3. When you are finished working with the dimension, choose "Enter" to complete the command.

Suppress

When you choose "suppress", the submenu of commands in the figure below will appear.

None

This command allows you to eliminate any previously defined suppression. Simply choose "None", and the command is completed. No other information is required. Your drawing will display all entities that comprise the current dimension.


Set suppression:

1st Witness

For best plotting results, no coincident lines should exist in your drawings. The "1st-Witness" command allows you to remove the first coincident entity.

1. Choose "1st-Witness".

2., Choose "Enter" to complete the command.

2nd-Witness

For best plotting results, no coincident lines should exist in your drawing. The "2nd-Witness" command removes the second coincident entity.

1. Choose 2nd-Witness.

2. Choose "Enter" to complete the command.

1st-Dimline

This command allows you to eliminate the first dimension line.

1. Choose "1st-Dimline.

2. Choose "Enter" to complete the command.

2nd-Dimline

This command allows you to eliminate the second dimension line.

1. Choose "2nd-Dimline".

2. Choose "Enter" to complete the command.

1st-Arrow

This command allows you to eliminate the first arrow.

1. Choose "1st-Arrow".

2. Choose "Enter" to complete the command.

2nd-Arrow

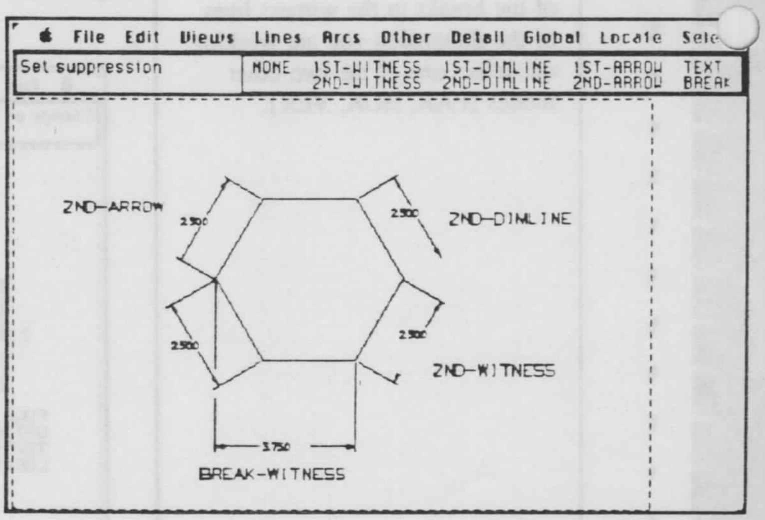
This command allows you to eliminate the second arrow.

1. Choose "2nd-Arrow".

2. Choose "Enter" to complete the command.

Text

This command allows you to eliminate all text in the current dimension. Choose "Text", then "Enter" and the text in the selected dimension will be eliminated.



Break

This command allows you to break witness lines for enhanced clarity in the drawing (see the figure on the previous page). To trim the selected witness line:

1. Choose "Break".

2. With the pointer, select two points on the witness line (the first closest to the geometry).

You can have up to three breaks in the same witness line when using the "Break" command. These breaks must be indicated successively away from the dimensioned element. To clear all of the breaks in the witness lines of the dimension you are working on, select one of the two other modes (PAR, HOR, VER).

Locate Submenu

Each time you choose "Locate", the submenu below will appear.

The commands in this submenu allow you to control the location of a selected dimension.

Auto-Center

Choosing this command toggles the automatic centering of your dimensioning text between witness lines.

Auto-Height

This command is not yet implemented.

Enter-Height

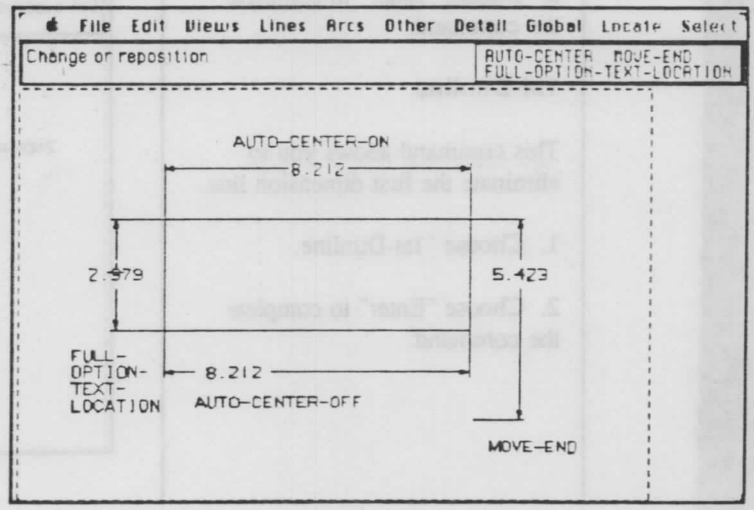
This command is not yet implemented.

Full-Option-Text-Location

This command allows you to enter a new location for text using the "Locate" menu.

Move-End

This command allows you to move a selected end of a dimension to a new location using the "Locate" menu. The system will automatically measure the distance between the endpoints and update the text onscreen.



Change or reposition

Smash

This command allows you to reduce a collection of geometry into individual entities. Each piece of geometry can then be modified independently of all others. By using the "Smash" command, the geometry no longer retains the associativity of a dimension; all that remains are individual pieces of geometry and text.

Boxed

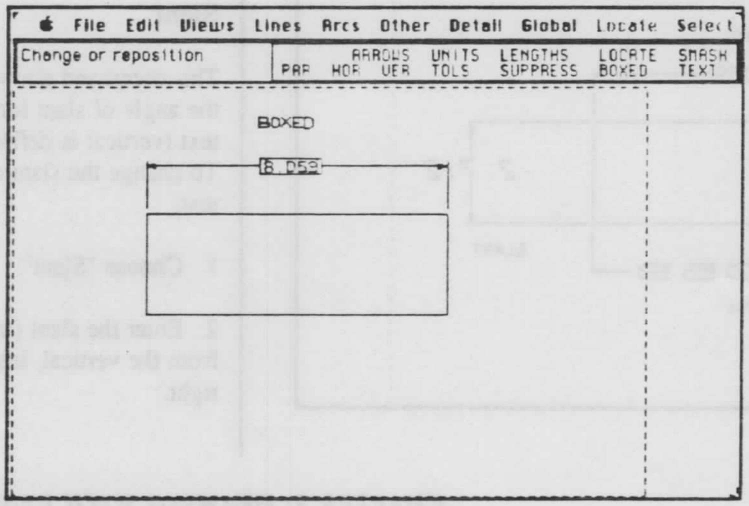
This command allows you to place a box around the dimension text. This command toggles between "Boxed" and "Unboxed".

Text Submenu

This submenu of commands allows you to modify the text of the current dimension (e.g. add text or change the quantitative value of the text).

Measured

This commands allows you to change the dimension text to the value of the true measurement of the geometry. The "Measure" command is useful if you have previously entered explicit values, but want to change back to the true values of the geometry.



Change or reposition

Entered

This command allows you to redefine the value of the current dimension by entering the value explicitly.

1. Choose "Entered".

2. Enter the desired value of the dimension.

3. Choose "Enter".

Add/Edit

This command allows you to add text to a dimension, and then edit the text.

1. Choose "Add/Edit".

2. Enter the desired text.

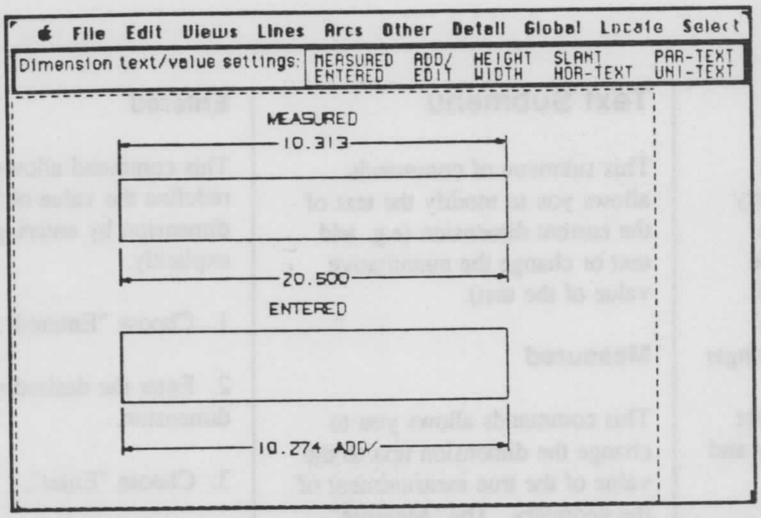
3. Choose "Enter".

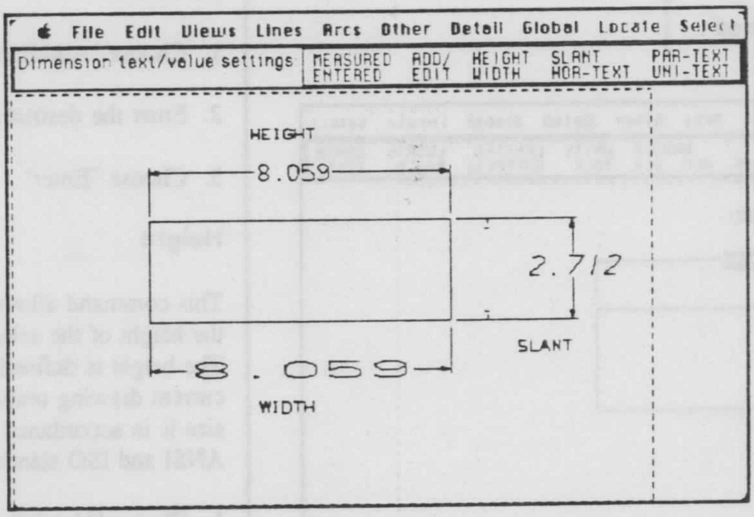
Height

This command allows you to set the height of the selected text. The height is defined in the current drawing units. The default size is in accordance with current ANSI and ISO standards.

1. Choose "Height".

2. Enter the desired height value.





3. Choose "Enter" twice (once to enter the value, the second to implement it).

Width

This command allows you to set the width of the letters comprising the text of the selected dimension. The width is entered as a ratio of height to width.

The initial default setting for width is 1 (e.g. .5 is thin, 2 is thick). To change the width of the text:

1. Select "Width".

2. Enter the desired width.

3. Choose "Enter", then "Enter" again to complete the modification.

Slant

This command allows you to set the angle of slant for the selectied text (vertical is defined as zero). To change the slant of the selected text:

1. Choose "Slant".

2. Enter the slant (in degrees) from the vertical, leaning to the right.

3. Choose "Enter", then "Enter" again to complete the modification.

If you enter negative values, the new slant will be to the left. (The initial default is to the vertical.)

Hor-Text

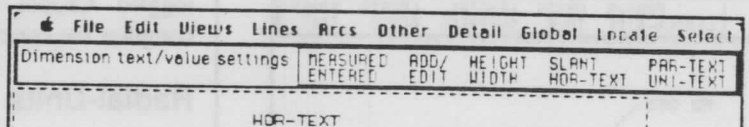
This command allows you to place text horizontally with respect to the selected dimension. ("Hor-Text" is the default setting for text orientation.) Simply choose "Hor-Text", and the text will automatically change to the new value.

Par-Text

This command allows you to place text parallel to the selected dimension lines. For a "Hor Linear" dimension, this command would be equivalent to the "Hor-Text" command. Simply choose "Par-Text", and the text will automatically change to the new value.

Uni-Text

When you select this command, text is automatically placed either horizontally or vertically, depending on the angle of the dimension witness lines.



Radial

This command allows you to create a radial dimension of an arc or circle

1. Select the arc or circle to be dimensioned. If Dynamic Dimensioning is On, the "Dynamic Dimensioning" submenu will appear.

2. Indicate a position for the text.

3. When you are finished working with that particular dimension, Choose "Enter" to complete the command.

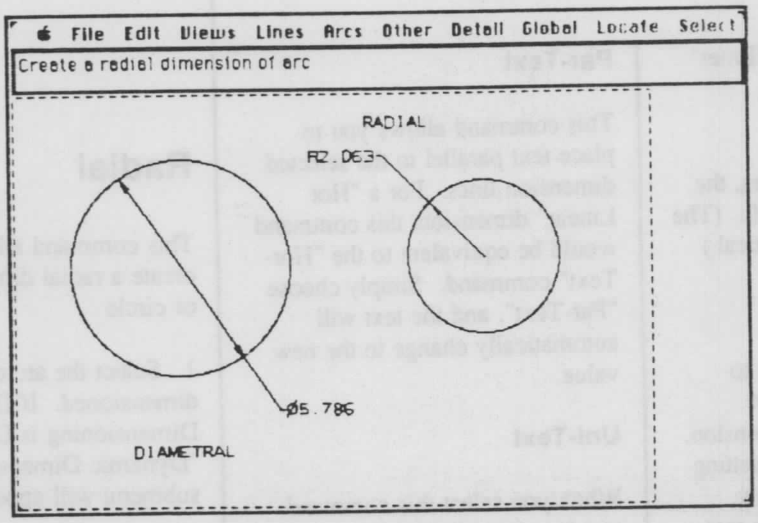
Diametral

This command allows you to create a diametral dimension of a circle .

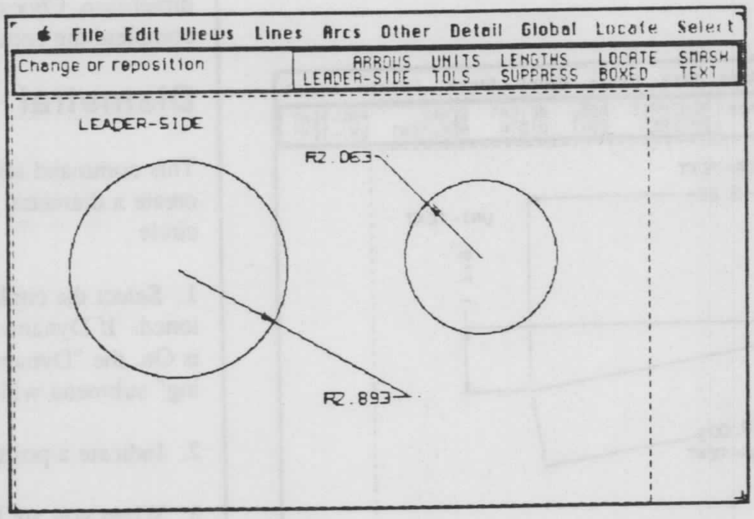
1. Select the circle to be dimensioned. If Dynamic Dimensioning is On, the "Dynamic Dimensioning" submenu will appear.

2. Indicate a position for the text.

3. When you are finished working with that particular dimension, Choose "Enter" to complete the command.



Create a radial dimension of arc



Change or reposition

Radial/Diametral Submenu

The "Dynamic Dimensioning" submenu for radial and diametral dimensions is very similar to the submenu for linear dimensions. Therefore, only the differences between the two menus will be described in this section.

The identical choices in the two menus are "Arrows", "Suppress", "Text", "Tols", "Smash", "Lenghths", and "Locate". For more information on these commands, see the "Linear Dimensioning" commands.

Leader-Side

This command allows you to change the side of the text from which the leader starts for the current dimension. Each time you select this command, the side is toggled. Choose "Enter" to complete the command.

Radial-Units

This command is just like the Linear-Units command in "Linear Dimensions". However, the radial and linear units are independent and can be set differently.

The "Angular" command allows you to create angular dimensions four ways. When creating angular dimensions, witness lines will appear at the end of the line closest to where you digitized.

1. Two lines - The lines can be defined by an existing line ("Line"), two points ("Pnt-Pnt"), or a point and an angle("Pnt-Angle").

2. The angle of an arc - ("Arc-Angle").

3. Two points and a center - ("Pnt-Pnt-Center").

4. Two points, the value of an angle, and the side that the center is on - ("Pnt-Pnt-Angle").

After the desired angular dimension is created, choose "Enter" to complete the command. If you choose "Line", "Pnt-Pnt", or "Pnt-Angle", then another line must also be selected by one of these methods. This additional menu is shown below.

Angular Submenu

The "Dynamic Dimensioning" submenu for angular dimensions will appear after you have dimensioned one or more angles while in the "Detail" menu. This menu is very similar to the "Linear Dynamic Dimensioning" submenu. Therefore, only the differences between the two menus will be described in this section.

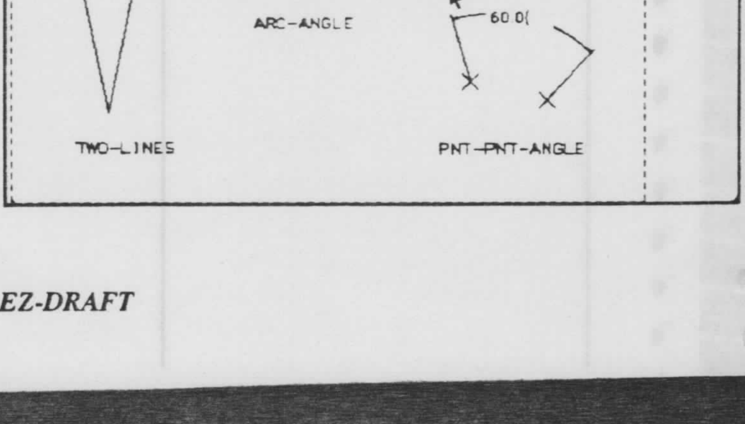
The identical choices are "Arrows", "Suppress", "Text", "Tols", "Smash", "Lenghths", and "Locate". For more information on these commands, see the "Linear Dimensioning"commands.

Break-Wit

This command is the same as the "Break-Wit" command in the "Suppress" submenu. For more information on the "Break-Wit" command, see "suppress" and "Break-Wit" in the "Linear Dimensioning" commands (also see Figure 8-28).

Angular-Units

When you choose "Angular-Units", the system will prompt you to choose either "Primary-Dimension" or "Tolerance". After your selection, these choices will appear in the submenu as shown below. This command allows


Create an angular dimension from

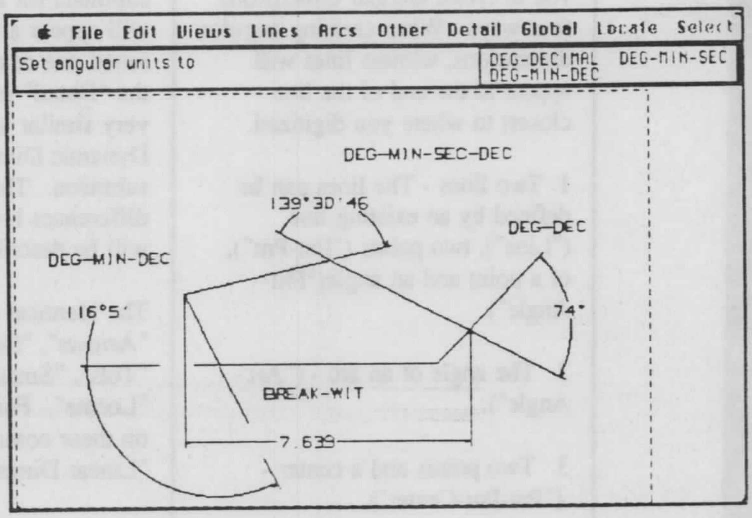
you to choose the submenu to modify angular dimensions from degrees to:

1. Degrees with a user specified number of decimal places "Deg-Decimal".

2. Degrees and minutes with a user specified number of decimal places "Deg-Min-Dec".

3. Degrees, minutes, and seconds with a user specified number of decimal palces "Deg-Min-Sec".

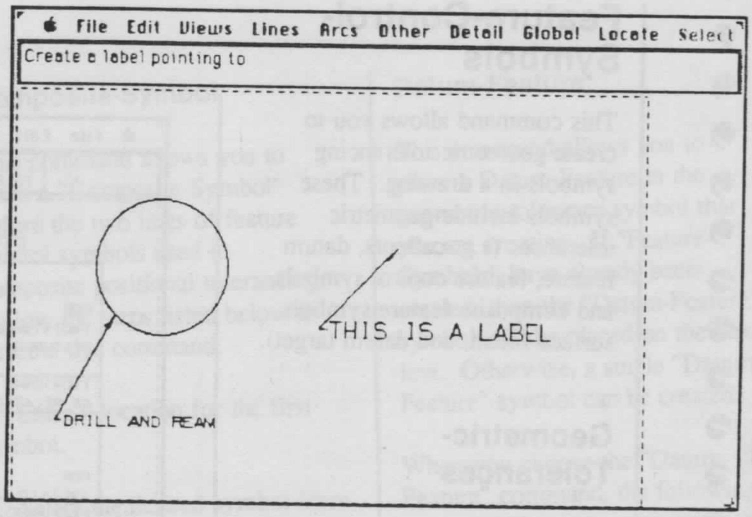
Simply make your selection, indicate the desired number of decimal places for the angular dimension, then choose "Enter". After you have finished working with that particular dimension, then choose "Return" to complete the command.



Set anguler units to

Label

The "Label" command allows you to place text anywhere in the current workview, with a leader line and arrow pointing to a selected spot on the drawing. This text can be modified or repositioned if Dynamic Dimensioning is On. After you place the text in the desired position onscreen, choose "Enter" to complete the command.



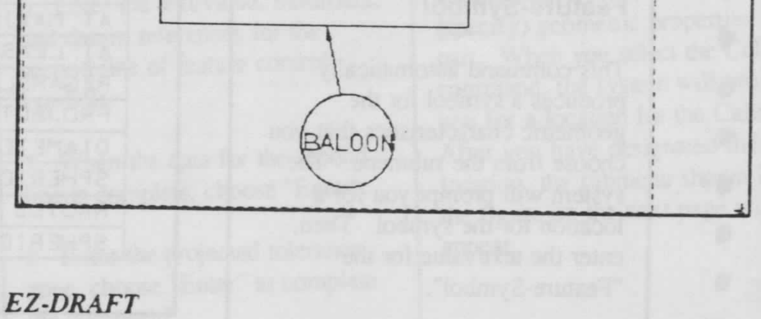
Create a label pointing to

Balloons

The "Balloons" command allows you to present text with a circle around it, and a leader line from the circle to a specified location. When you select this command, the system will prompt you for a starting position, a location for the text, and then the text.

The "Balloons" command uses a default radius based on the number of characters entered. This can be changed after the balloon is first placed by changing the "Wit-Line-Ext" under "Lengths" in the "Balloons" editing submenu.

If you change the size of the text, the size of the balloon will not change. To change the balloon size, use the "Wit-Line-Ext" command in the "Lengths" submenu.



Create a balloon pointing to

Feature-Control-Symbols

This command allows you to create geometric tolerancing symbols in a drawing. These symbols include geometric tolerances (e.g. callouts, datum feature, feature control symbols, and composite feature symbols, surface finish, and datum target).

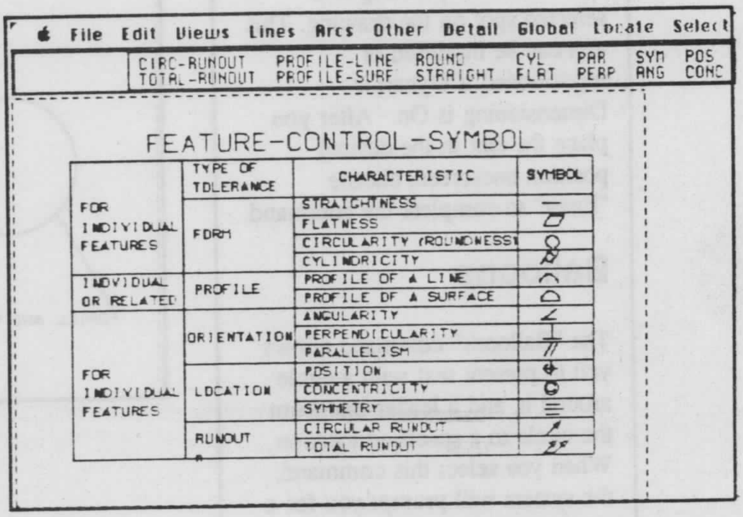
Geometric-Tolerances

When selected, this command allows you to create geometric tolerancing symbols and place them in your drawing. With the "Geometric-Tolerances" command, you can create callouts, datum feature, feature control symbols, and composite feature symbols.

After you select a geometric tolerancing symbol to create, the system will prompt you for a location for the symbol.

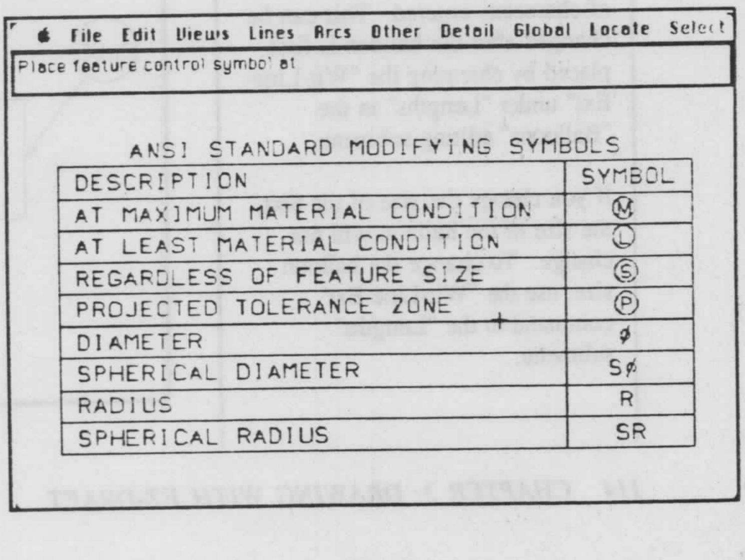
Feature-Symbol

This command automatically produces a symbol for the geometric characteristics that you choose from the submenu. The system will prompt you for a location for the symbol. Then, enter the text/value for the "Feature-Symbol".



FEATURE-CONTROL-SYMBOL

	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL
FOR INDIVIDUAL FEATURES	FORM	STRAIGHTNESS	⏤
		FLATNESS	⏥
		CIRCULARITY (ROUNDNESS)	○
		CYLINDRICITY	⌭
INDVIDUAL OR RELATED	PROFILE	PROFILE OF A LINE	⌒
		PROFILE OF A SURFACE	⌓
FOR INDIVIDUAL FEATURES	ORIENTATION	ANGULARITY	∠
		PERPENDICULARITY	⊥
		PARALLELISM	//
	LOCATION	POSITION	⌖
		CONCENTRICITY	◎
		SYMMETRY	⌯
	RUNOUT	CIRCULAR RUNOUT	↗
		TOTAL RUNOUT	⌰



Place feature control symbol at

ANSI STANDARD MODIFYING SYMBOLS

DESCRIPTION	SYMBOL
AT MAXIMUM MATERIAL CONDITION	Ⓜ
AT LEAST MATERIAL CONDITION	Ⓛ
REGARDLESS OF FEATURE SIZE	Ⓢ
PROJECTED TOLERANCE ZONE	Ⓟ
DIAMETER	⌀
SPHERICAL DIAMETER	S⌀
RADIUS	R
SPHERICAL RADIUS	SR

After choosing the desired "Feature-Symbol" a submenu will prompt you to choose a modifier MAX- MATERIAL, LEAST-MATERIAL or REGARDLESS-OF-SIZE. If you do not want a modifier, choose "Enter". At this point, the system will prompt you for a datum reference, and then a modifier (if desired).

The table on the bottom of the previous page shows the ANSI standard modifying symbols. The modifiers "Diameter", "Spherical Diameters", "Radius" and "Spherical Radius" are text characters that you can enter directly from the keyboard.

This procedure of entering datum references, tolerances, and modifiers will continue until you choose "Enter". When you choose "Enter", The following prompt will appear: "Do you want a projected tolerance zone frame?". If No, "Return", then the first Geometric-Tolerance menu will appear. If you choose "Yes", "Enter", then this prompt will appear: "Projected tolerance zone value is". Simply enter the desired value, then choose "Enter" or "Return" to complete the command. At this point, you will be returned to the first Geometric-Tolerance menu.

Composite-Symbol

This command allows you to place a "Composite-Symbol" before the two lines of feature control symbols used in composite positional tolerancing. Follow the steps listed below to execute this command.

1. Enter a location for the first symbol.

2. Select the desired symbol from the menu of "Feature-Symbols".

3. Enter the text/value for the first line of "Feature Symbols".

4. Enter the datum references and modifier (as in "Feature-Symbol").

5. When you have completed the first two steps and the data for the first line is complete, choose "Enter".

6. Enter the text/value, modifiers, and datum references for the second line of feature control symbols.

7. When the data for the second line is complete, choose "Enter".

8. Enter the projected tolerance zone, choose "Enter" to complete the command.

Datum-Feature

This command allows you to place a Datum-Feature in the geometric tolerance symbol that you are creating. If "Feature-Symbols" have already been created, then the "Datum-Feature" symbol will be placed on the next line. Otherwise, a single "Datum-Feature" symbol can be created.

When you choose the "Datum-Feature" command, the following prompt will appear, requesting the datum identifying letter, "Continue with datum feature consisting of". Once you have entered the text, choose "Enter or "Return"to complete the command.

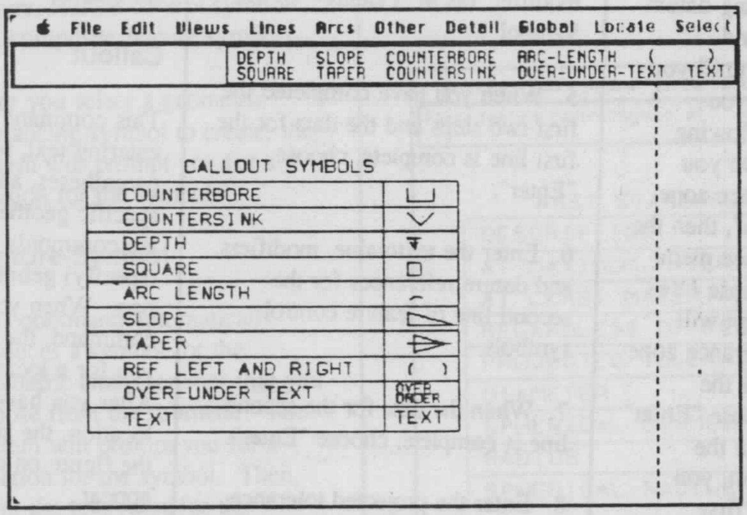
Callout

This command can be used for entering text, referencing parentheses, and for placing specific geometric symbols that are commonly used to Callout (specify) geometric properties of a part. When you select the Callout command, the system will prompt you for a location for the Callout. After you have designated the location, the submenu shown in the figure on the next page will appear.

The options "Arc-Length", "Over-Under-Text", and "Text" will prompt you for text input. It is important to remember that the following keys on the keyboard have special functions.

- The "{" key is a symbol for degrees.
- The "}" key is a minute mark (e.g. angular degrees and minutes).
- The "|" key is a diameter symbol.
- The "\" key is a plus/minus symbol.

Simply select the desired Callout symbols and they will appear onscreen beginning at the location you have entered. When you are finished selecting Callouts, choose "Return" to return on the first Geometric-Tolerances menu.


File Edit Views Lines Arcs Other Detail Global Locate Select
DEPTH SLOPE COUNTERBORE ARC-LENGTH ( )
SQUARE TAPER COUNTERSINK OVER-UNDER-TEXT TEXT
CALLOUT SYMBOLS
COUNTERBORE
COUNTERSINK
DEPTH
SQUARE
ARC LENGTH
SLOPE
TAPER
REF LEFT AND RIGHT
( )
OVER-UNDER TEXT
OVER UNDER
TEXT
TEXT

Placement

The commands in this submenu allow you to place the completed geometric tolerancing symbol in the desired location within your drawing.

You can place a symbol by selecting the Move-Symbol-Location command. The symbol can be moved by indicating one of the four "corners" as a point of location for the symbol to be moved. (See additional submenu below). This command can be used more than once if necessary.

The Leader-Left and Leader-Right commands allow you to place a Leader line pointing from the symbol you have created to the desired element. When the symbol and leader line are in the desired location, Choose "Enter" to complete the command.

Surface-Finish-Marks

This command allows you to create Surface-Finish symbols. You can indicate properties such as roughness height, removal by machining allowances, and roughness width.

To create Surface-Finish-Marks:

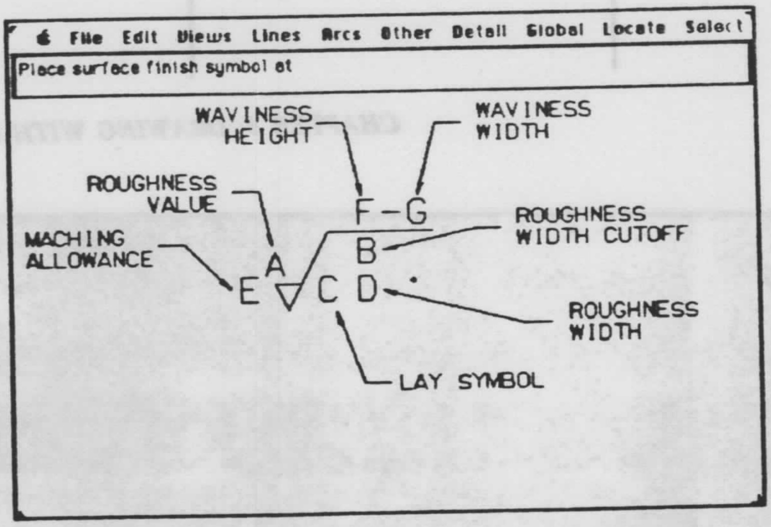
1. Choose "Surface-Finish-Marks".

2. Select a location for the surface finish symbol.

3. You will be prompted to choose a roughness specification height (SINGLE-VALUE, MIN/MAX, or N-SERIES), then enter a value(s) for this height (choose "Return" for no roughness height).

4. Indicate either OBLIGATORY (CLOSED), PROHIBITED (CIRCLE), OPTIONAL (OPEN) or "Enter" for no indication for removal method..

5. A menu with the selections from the figure below will now appear. Choose one of the options displayed. If you choose "Return" at this time, the symbol will be created and no other changes or additions can be made. However, if you want to place additional "Surface-Finish-Marks" in the symbol, simply continue to select the desired options from the menu displayed. After you have selected an option, enter the the desired value(s).



Place surface finish symbol at

Lay-Symbol

"Lay-Symbols" indicate the directional pattern of the surface texture. When you choose this option, the following sub menu selections will appear:

1. MULTI-DIR(M)

2. PAR(=)

3. PERP

4. CIRC(C)

5. ANG(A)

6. RADIAL(R)

7. PITTED(P)

If you choose "Return" at this time, the symbol will be created and no other changes or additions can be made. However, if you want to place additional "Surface-Finish-Marks" in the symbol, continue to select the desired options from the menu displayed.

Roughness-Width

This option allows you to enter a value for the "Roughness-Width". The "Roughness-Width" is the distance between successive peaks or ridges that compose the predominate pattern of roughness on a surface. Simply choose this option, then enter the desired roughness width cutoff value.

If you choose "Return" at this time, the symbol will be created and no other changes or additions can be made. However, if you want to place additional "Surface-Finish-Marks" in the symbol, simply continue to select the desired options from the menu displayed.

Width-Cutoff

Roughness "Width-Cutoff" is the greatest spacing of repetitive surface irregularities included in the measurement of roughness height. This value must be greater than the "Roughness-Width". Enter the value for the roughness "Width-Cutoff".

If you choose "Return" at this time, the symbol will be created and no other changes or additions can be made. However, if you want to place additional "Surface-Finish-Marks" in the symbol, simply continue to select the desired options from the menu.

Waviness

"Waviness" is the (generally) widely spaced component of the surface texture. It is usually more widely spaced than the roughness "Width-Cutoff". You will be prompted to enter the waviness height (and width) values.

If you choose "Return" at this time, the symbol will be created and no other changes or additions can be made. However, if you want to place additional "Surface-Finish-Marks" in the symbol, simply continue to select the desired options from the menu.

Machining-Allowance

When you choose this option you will be prompted to enter the desired value for the "Machining-Allowance".

If you choose "Return" at this time, the symbol will be created and no other changes or additions can be made. However, if you want to place additional "Surface-Finish-Marks" in the symbol, simply continue to select the desired options from the menu.

Datum-Target

This command allows you to create "Datum-Target" symbols in your drawing. Datum targets can be created with different target points, datum letters, target numbers and target areas.

To create a Datum-Target symbol:

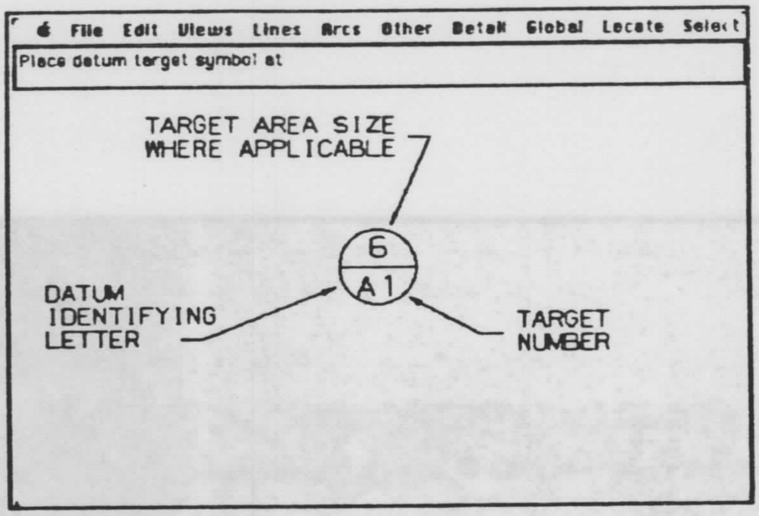
1. Choose "Datum-Target".

2. Select a location for the Datum-Target symbol.

3. Indicate where the Datum-Target symbol will point. Now choose from X-FOR-TARGET-POINT, +-FOR-TARGET-POINT or NO-TARGET-POINT. You may also choose NO-LEADER-LINE if it is not needed.

4. The next prompt that appears will request a datum letter and a target number. Enter these, then choose "Enter".

The next menu that appears contains commands that allow you to enter additional "Datum-Target" symbols to the first symbol. Simply choose LEFT, RIGHT, ABOVE or BELOW to place the additional "Datum-Targets". This process will continue until you choose "Return" when the prompt, "Attach datum target symbol to" appears again.


Place datum target symbol at

File Edit Views Lines Arcs Other Detail Global Locate Select
TARGET AREA SIZE WHERE APPLICABLE
6
A1
DATUM IDENTIFYING LETTER
TARGET NUMBER

Dynamic Dim On-Off

The "Dynamic Dim On-Off" command toggles the dynamic editing of dimensions. Dynamic Dimensioning allows you interactively change the characteristics of the dimension you are working with.

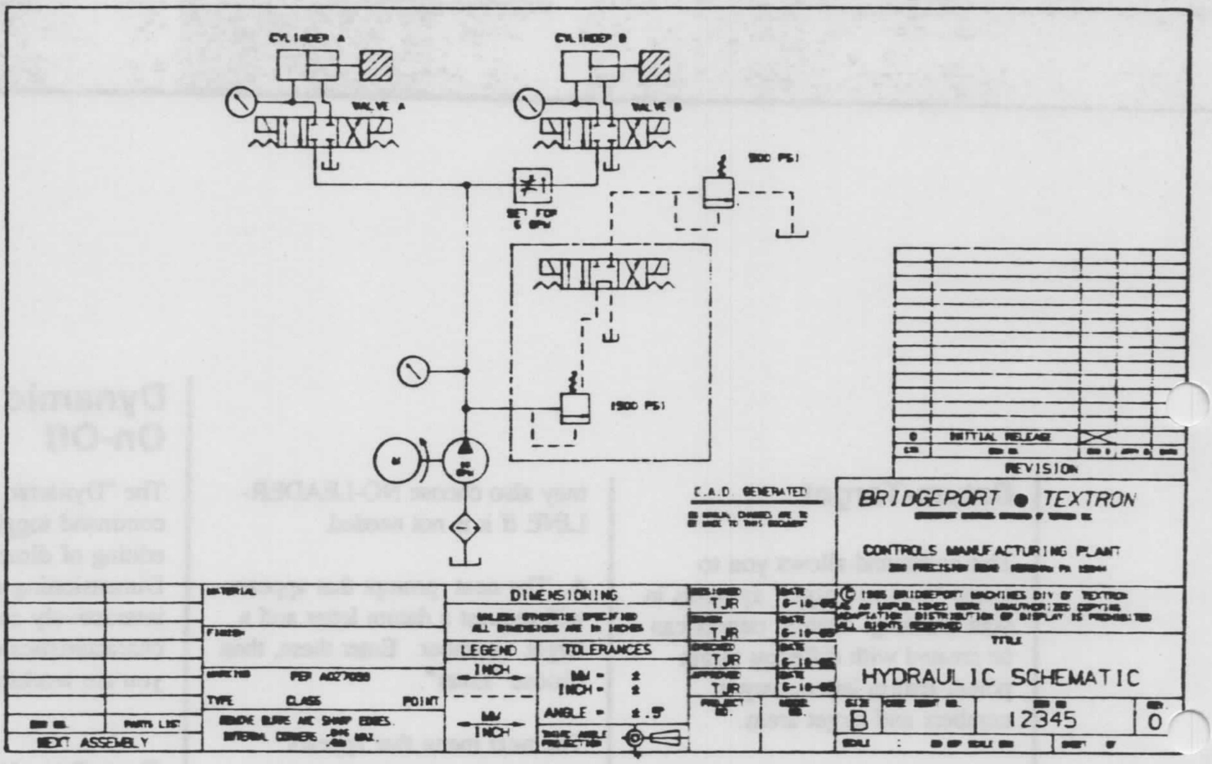
Set Similar To Dim

This command sets dimension characteristics similar to an existing dimension such that subsequent dimensions will have the same characteristics.

1. Choose "Set Similar To Dim"

2. Indicate the dimension with the desired dimension characteristics.

Chapter 4



Making Changes and Finishing Up

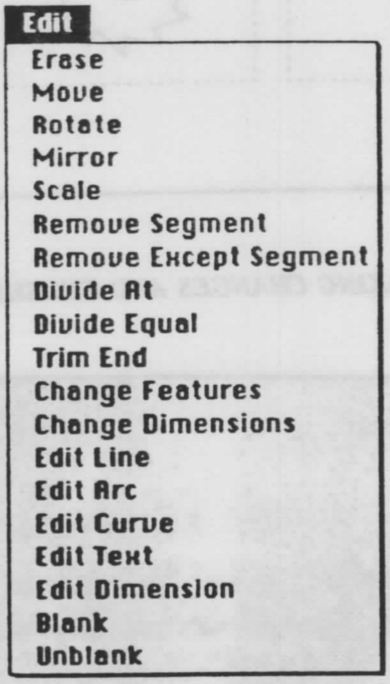
After completing a drawing with EZ-Draft you may want to make some changes or file the drawing and move on. The purpose of this chapter is to describe the menus and commands that enable you to make changes to your drawings as well as how to access and write to drawing files.

Edit

The 19 selections in the "Edit" menu allow you to make changes to your drawing. The "Move", "Rotate", "Mirror" and "Scale" commands also allow you to use previously drawn geometry to create new geometry.

Erase

This command allows you to erase elements from your drawing. To erase an element(s), simply select the element(s) with the multiple selection methods that appear under the "Select" menu, then choose "Enter", and the elements are permanently removed.



Move

This command allows you to move or translate a group of selected elements from their current location to a new location.

1. Select the elements to be moved.

2. Enter the point location to move from.

3. Enter the point location to move to. The actual distance and direction that the selected geometry will be moved is determined by the difference between the "from" and "to" points.

4. Enter the number of copies. If the number of copies is 0, then the elements are moved. If the number of copies is 1, then a single copy of the geometry is created at the specified location leaving the original elements at their original location. This location is determined by taking the distance and direction calculated by the "from" and "to" points, and applying it to the current position of the selected geometry. If the number of copies is greater than 1, then each additional copy is created at consecutive distances from the original until completed.

Rotate

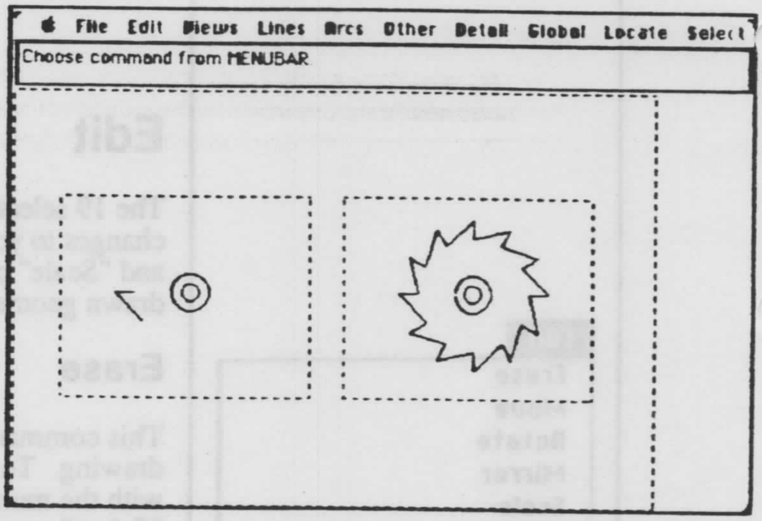
This command allows you to rotate a group of selected elements through an entered angle about an entered point.

1. Select the elements to be rotated.

2. Enter the angle of rotation.

3. Enter the point about which the geometry is to be rotated. The geometry is rotated from its current position through the entered angle about the entered point to its new position.

4. Enter the number of copies desired. If 0 is selected, then the original elements are moved. If the number is 1, a single copy of the geometry is created, rotated through the entered angle about the entered point leaving the original elements unchanged. If the number of copies is greater than 1, then each additional copy is created at consecutive incremental angles of the entered angle from the original about the entered point until completed leaving the original elements unchanged.



Choose command from MENUBAR.

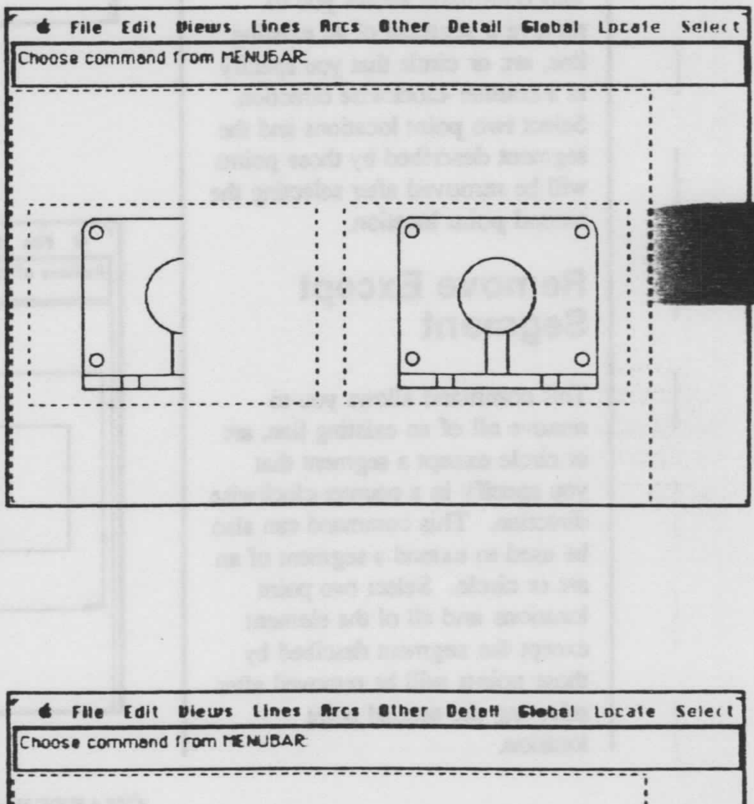
Mirror

This command allows you to create a mirror image of a group of selected elements about a line defined by two points.

1. Select the elements to be mirrored.

2. Enter the two points that will define the line about which the mirror will occur. The selected geometry is then mirrored about the line.

3. Entering 0 allows you to create a single mirror image (the original elements are moved). Entering 1 creates a copy of the selected elements in the new position, leaving the original elements unchanged.



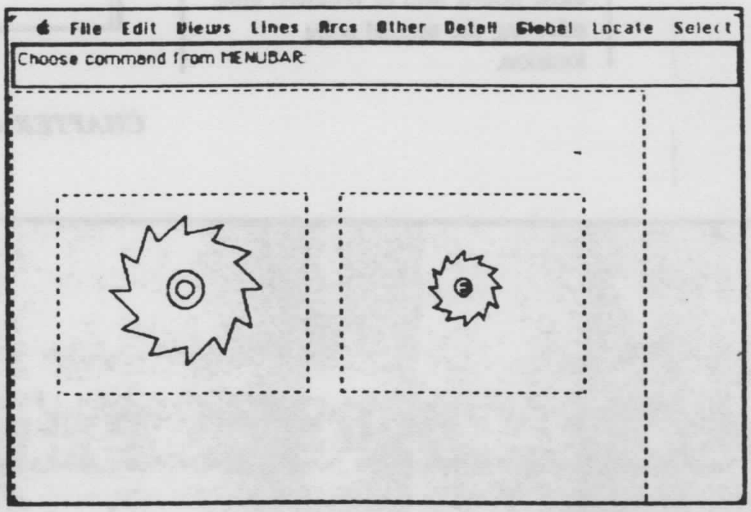
Choose command from MENUBAR:

Scale

This command allows you to scale a group of selected elements by an entered scale factor about an entered point.

1. Select the elements to be scaled.

2. Enter the point about which the geometry is to be scaled. The selected geometry is then scaled about the entered point to its new size.



Choose command from MENUBAR:

3. Enter the scale factor for the horizontal (X) and then vertical (Y) direction.

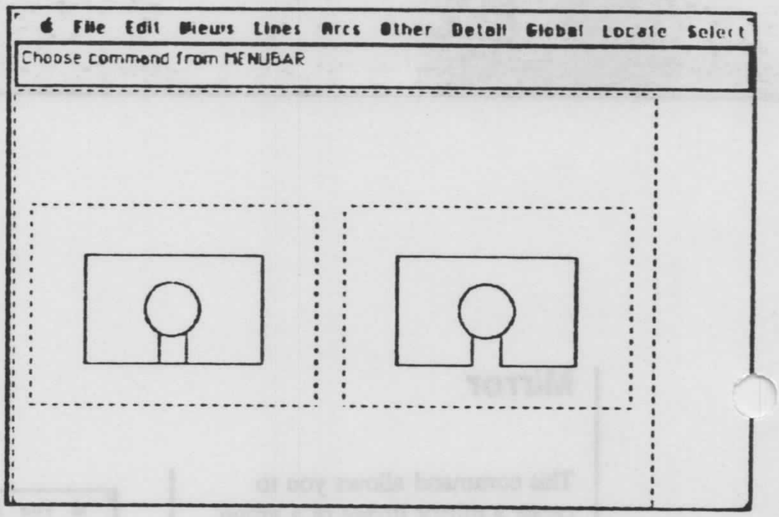
4. Enter the number of copies desired. If 0 is selected, the original elements are moved to the new location at their new size. If 1 is selected, a copy of the selected geometry is then made, scaled about the entered point to its new size.

Remove Segment

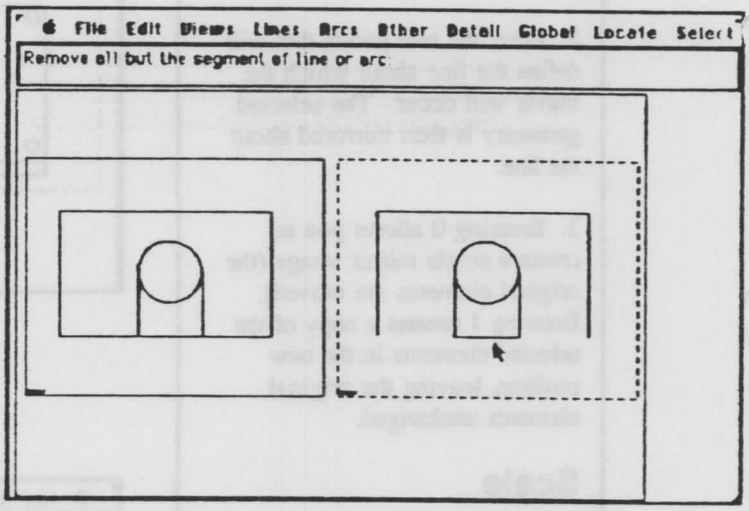
This command allows you to remove a segment of an existing line, arc or circle that you specify in a counter-clockwise direction. Select two point locations and the segment described by those points will be removed after selecting the second point location.

Remove Except Segment

This command allows you to remove all of an existing line, arc or circle except a segment that you specify in a counter-clockwise direction. This command can also be used to extend a segment of an arc or circle. Select two point locations and all of the element except the segment descibed by those points will be removed after selecting the second point location.



Choose command from MENUBAR



Remove all but the segment of line or arc

Divide At

This command allows you to divide a line or arc into two lines.

1. Select a line or arc to be divided.

2. Enter the point where you want the line or arc to be broken. If the point is not on the line, the perpendicular drop from the entered point is used to determine the break point. The two lines can now be treated as any other lines.

Divide Equal

This command divides a selected line or arc into a specified number of equal length lines.

1. Enter the number of pieces into which the existing line or arc will be divided.

2. Select the line or arc to be divided. Each of the new pieces can now be treated as any other lines or arcs.

Trim End

This command allows you to trim or extend a line or an arc to a point location.

1. Select the line or arc to be trimmed or extended. The nearest endpoint during selection is the end that will be trimmed or extended.

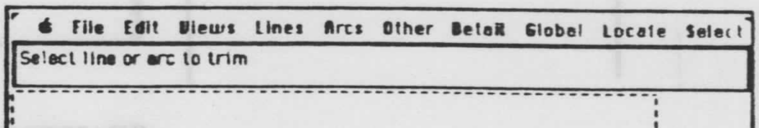
2. Enter the point to be trimmed or extended to.

Change Features

This menu contains commands that allow you to change the features or display characteristics of selected elements. For example, the "Layer", "Color", "Font", or "Group-name" of a specific element can be modified or changed.

Layer

This command allows you to change the "transparent" layer on which the selected element is located.



Select line or arc to trim

Color

This command allows you to change the color that is used to display or plot an element(s).

Curve-Font

This command allows you to change the font of lines, arcs, and b-splines.

Group

This command allows you to change the Group-name with which this and other elements are referenced.

Line-Weight

This command is not implemented yet.

Change Dimensions

This command allows you to change the sizes of previously created dimensions. To execute this command, simply select the dimension(s) that you wish to change, select a new setting from the choices displayed, then choose "Enter" to complete the command. Choose ARROWS, TOLS, LENGTHS, BALOON-RAD, ANG-RAD-LIN-UNITS, SUPRESS, BOXED or TEXT from the displayed submenu.

Edit Line

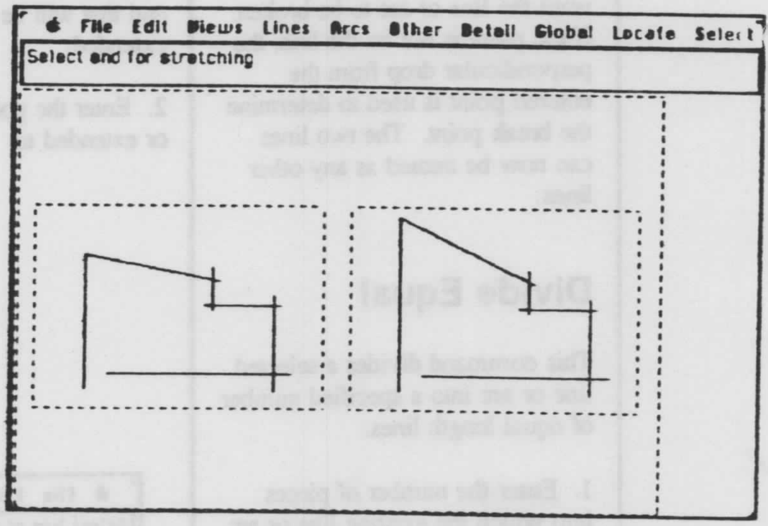
This submenu contains commands that allow you to edit an individual line.

Stretch-End

This command allows you to move the endpoint of selected line(s). The system checks for lines with common endpoints and then moves all of their endpoints to the new position.

1. Select the endpoint to be repositioned.

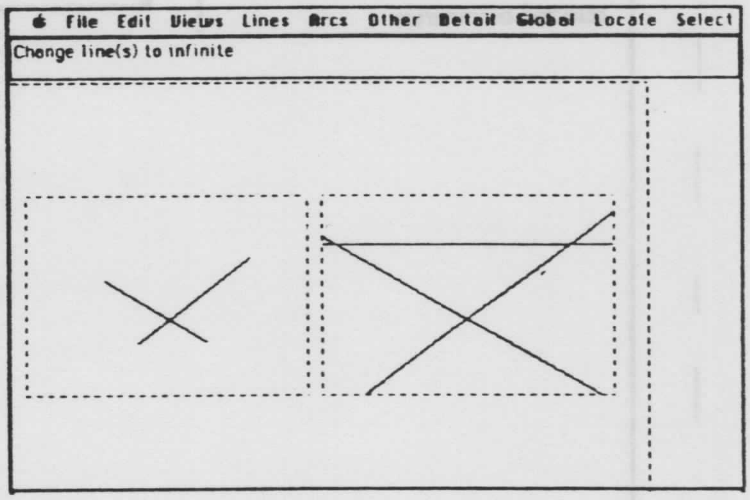
2. Enter the new location for the endpoint.



Select end for stretching

Infinite-Line

This command allows you to extend a selected line to the extent of its view (i.e. converting a line into a construction line).


File Edit Views Lines Arcs Other Detail Global Locate Select

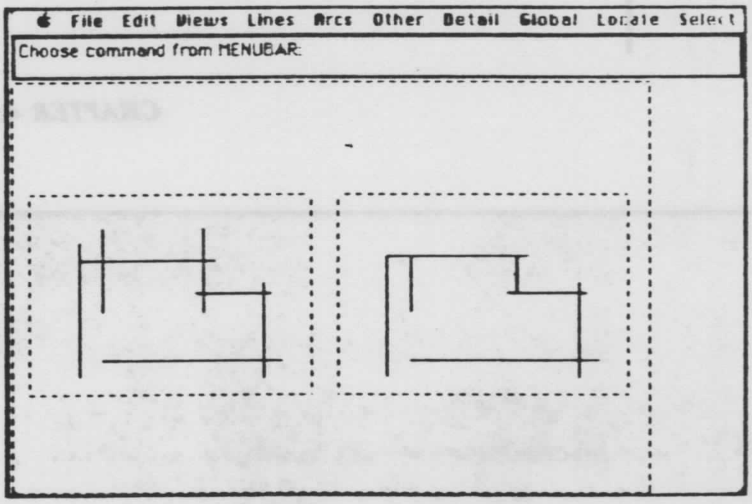
Change line(s) to infinite

Trim-To-One-Bound

This command allows you to trim or extend one or more lines to their intersection with the selected boundary.

1. Select the line to be used as the boundary.

2. Select the line or lines to be trimmed or extended. The side of the boundary on which you select the line(s) is the side that will be trimmed.


File Edit Views Lines Arcs Other Detail Global Locate Select

Choose command from MENUBAR:

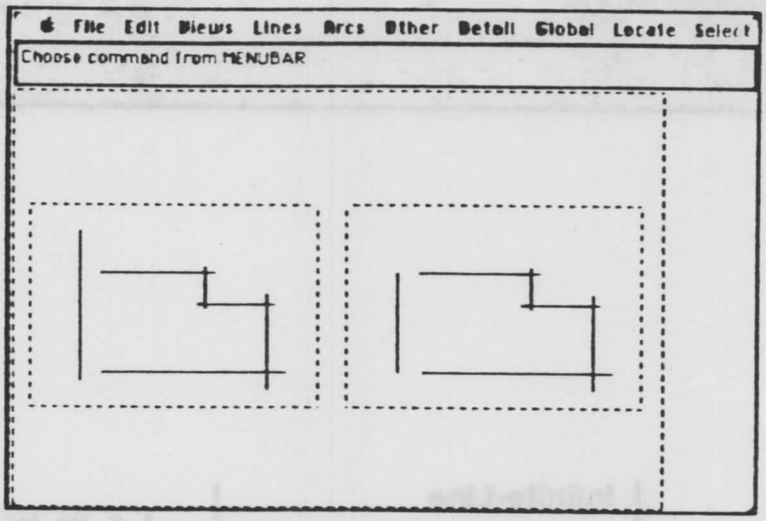
Trim-To-Two-Bounds

This command allows you to trim or extend one or more lines to their intersections with the selected boundaries.

1. Select the line to be used as the first boundary.

2. Select the line to be used as the second boundary.

3. Finally, select the lines to be trimmed. The lines you select will be trimmed between the two selected boundaries.



Choose command from MENUBAR

Edit-Arc

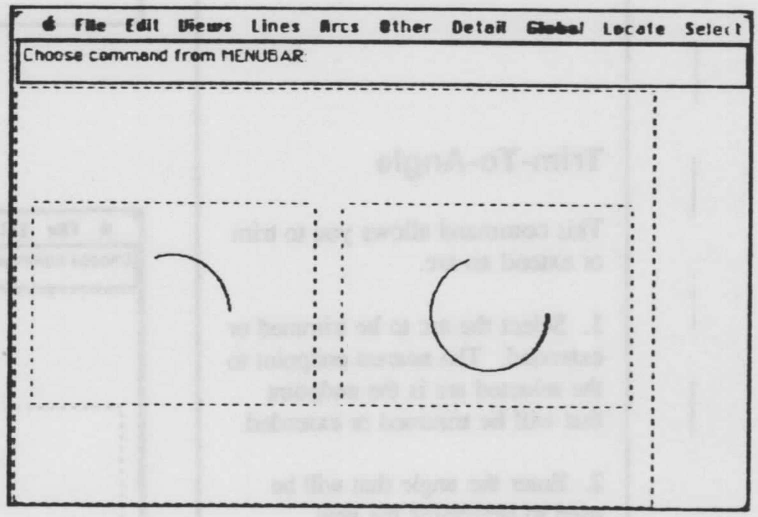
The "Arc/Circle" submenu contains commands to edit an individual arc or Circle (e.g. changing radius or diameter, and trimming an arc to a point or angle).

Complement-Arc

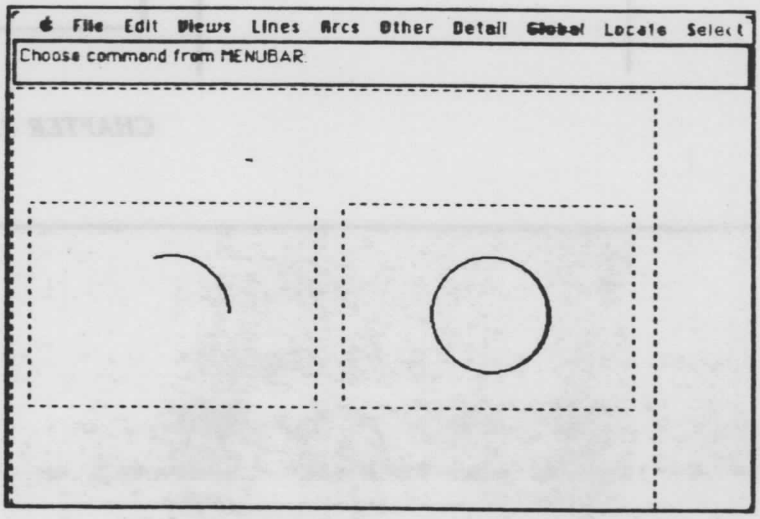
This command yields the complement of a selected arc. Select the arc to be complemented. The system will automatically convert the arc to its complement. If a circle is selected, the circle will be erased (the complement of a circle is no circle).

Arc-To-Circle

This command allows you to convert one or more arcs into full circles. Select the arc(s) to be converted into circles. The system will automatically convert the arcs into circles.



Choose command from MENUBAR.



Choose command from MENUBAR.

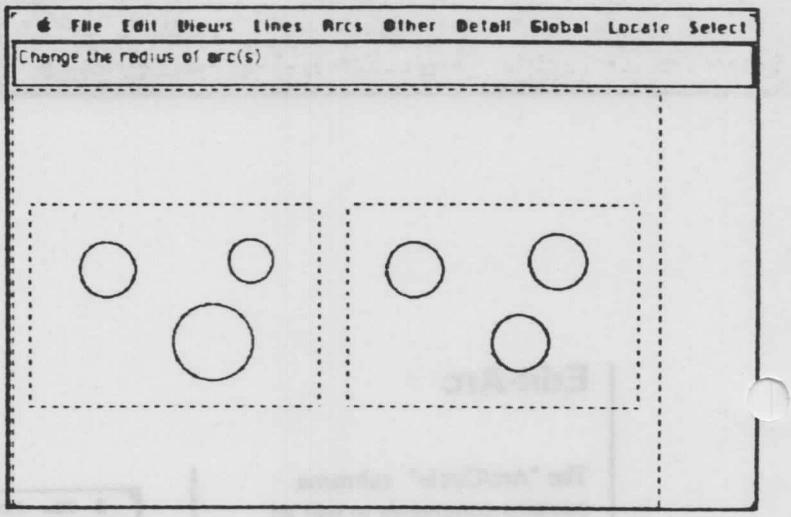
Change-Radius

This command changes the radius of one or more selected arcs or circles from their current radius to a newly specified radius.

1. Select the arc(s) or circle(s) to be changed.

2. Enter the new radius.

The system will automatically change the radius of the selected arc(s) or circle (s).


File Edit Views Lines Arcs Other Detail Global Locate Select

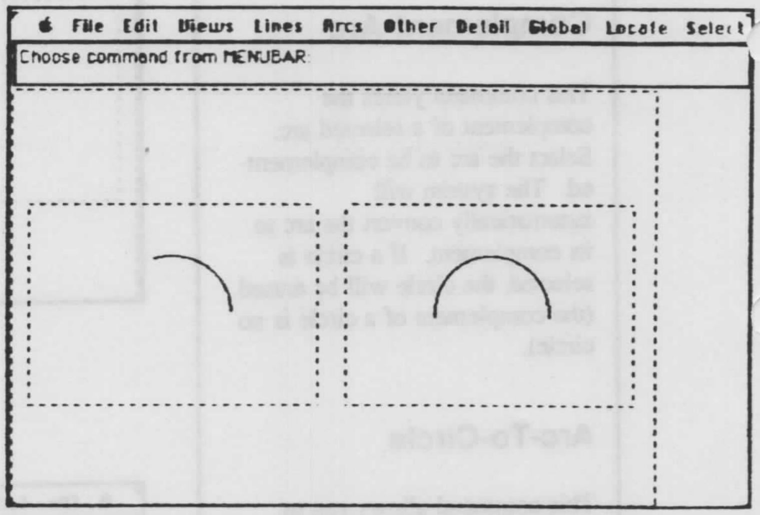
Change the radius of arc(s)

Trim-To-Angle

This command allows you to trim or extend an arc.

1. Select the arc to be trimmed or extended. The nearest endpoint to the selected arc is the endpoint that will be trimmed or extended.

2. Enter the angle that will be used to determine the new endpoint of the arc.


File Edit Views Lines Arcs Other Detail Global Locate Select

Choose command from MENUBAR:

Edit-Curve

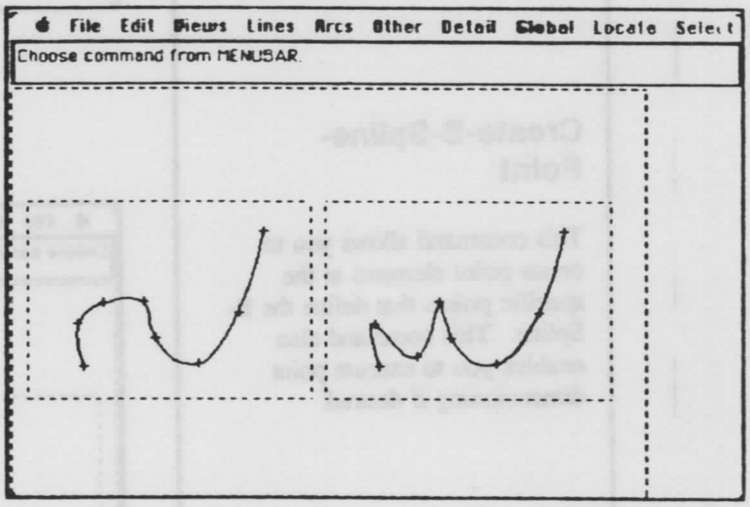
The "Curve" submenu contains commands that allow you to edit B-Spline curves (e.g. move, add, or delete points which make up the B-Spline).

Move-B-Spline-Point

This command allows you to move a B-Spline point.

1. Select a point from the displayed points to be moved.

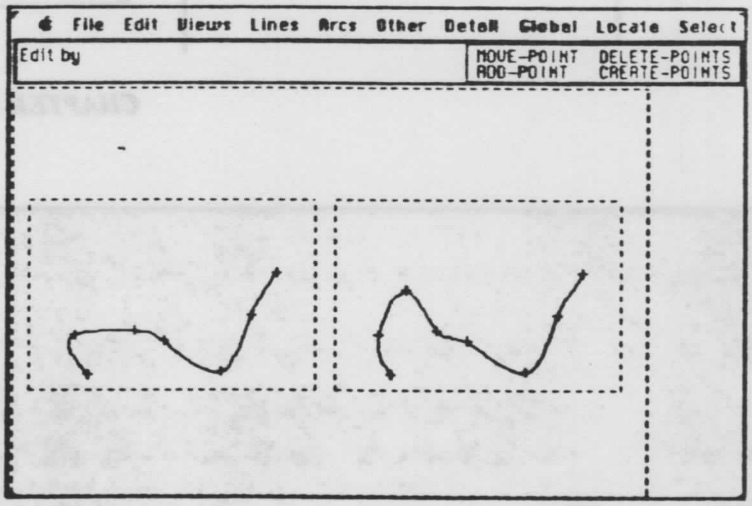
2. Enter the new location of the B-Spline point.



Choose command from MENUBAR.

Add-B-Spline-Point

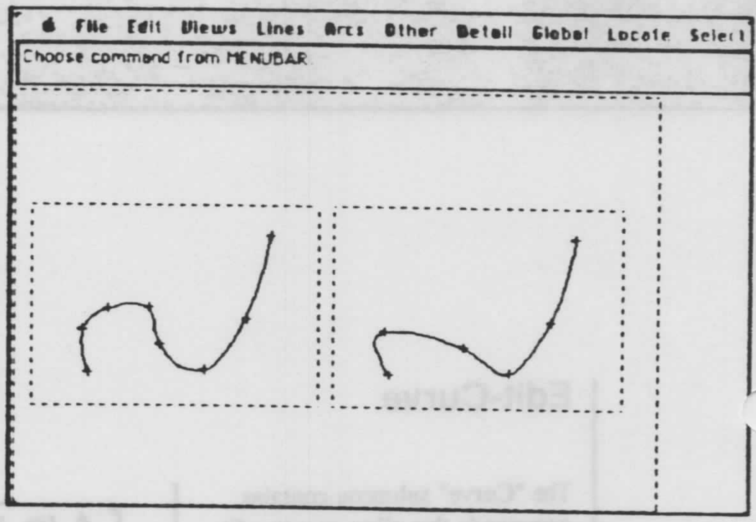
This command allows you to add a B-Spline point. Simply select the two B-Spline points between which the new point will be added. If you select the same endpoint twice, then a point will be added to extend the B-Spline.



Edit by

Delete B-Spline Point

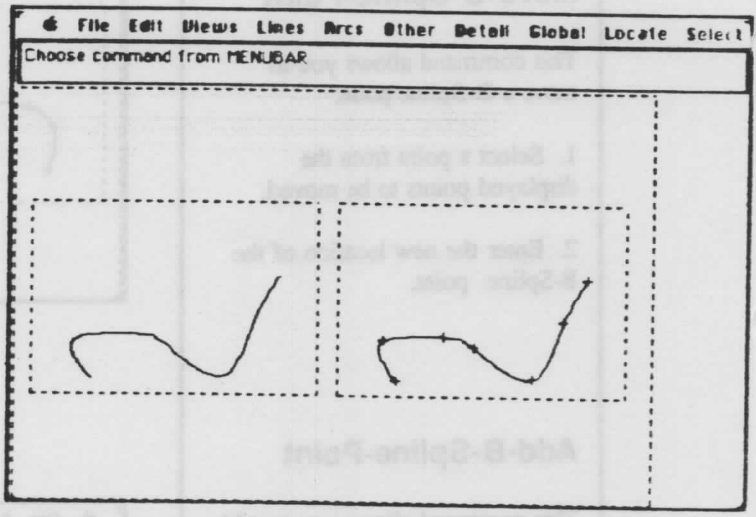
This command allows you to delete as many B-Spline points as you select. Simply indicate which B-Spline point you want to delete. The system will automatically delete the indicated points. Note that there must be at least four points in any B-Spline .



Choose command from MENUBAR

Create-B-Spline-Point

This command allows you to create point elements at the specific points that define the B-Spline. This command also enables you to execute point dimensioning if desired.



Choose command from MENUBAR

Edit-Text

Choosing this command allows you to edit previously written text in any manner. The submenu under "Edit Text" is identical to the submenu that appears immediately after entering the first line of text in the Detail menu. When you choose "Edit Text" and indicate the text for editing, the submenu appears. For a full description of the commands in the submenu, see "Notes" in the Detail section of Chapter 3.

Edit-Dimension

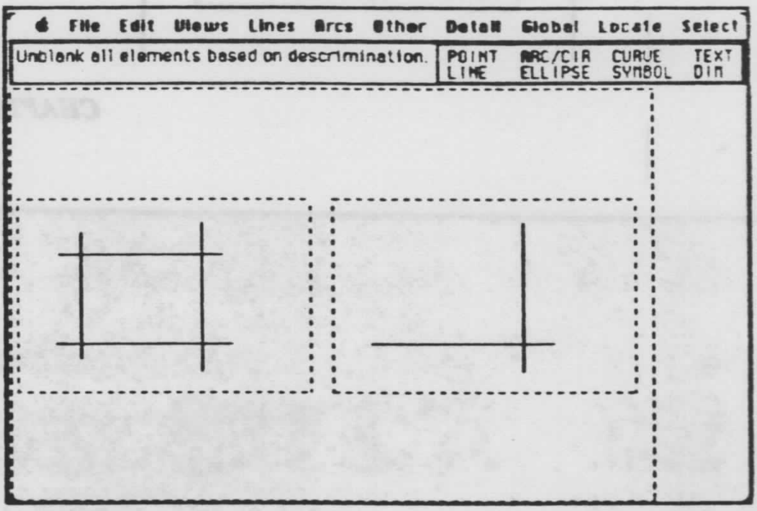
The "Edit-Dimension" command allows you to select a specific dimension to be edited or moved. Once you select this command, you can make further selections from the "Dynamic Dimensioning" submenus. (in essence, this command returns you to "Dynamic Dimensioning"). This submenu is identical to the editing options available immediately after creating a dimension. See the "Detail" menu in Chapter 3 for more information.

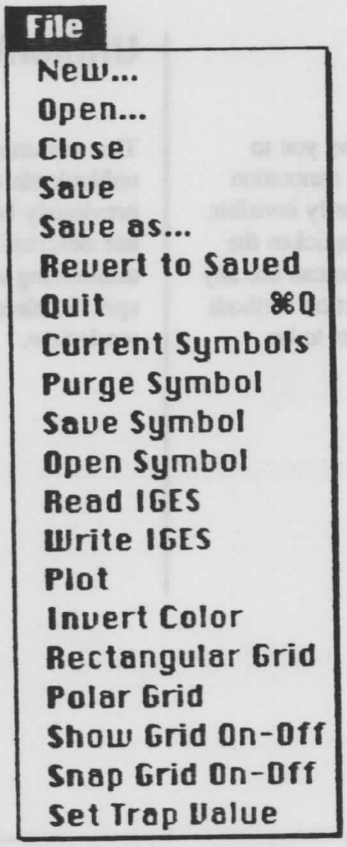
Blank

This command allows you to select geometry and annotation and make it temporarily invisible. Using "Blank" will quicken the redraw process. You can use any of the multiple selection methods to select the elements to be blanked.

Unblank

This command allows you to unblank elements which have previously been blanked. You can use descrimination when unblanking in order to unblank specific elements in the current workview.


Unblank all elements based on descrimination.



File Menu

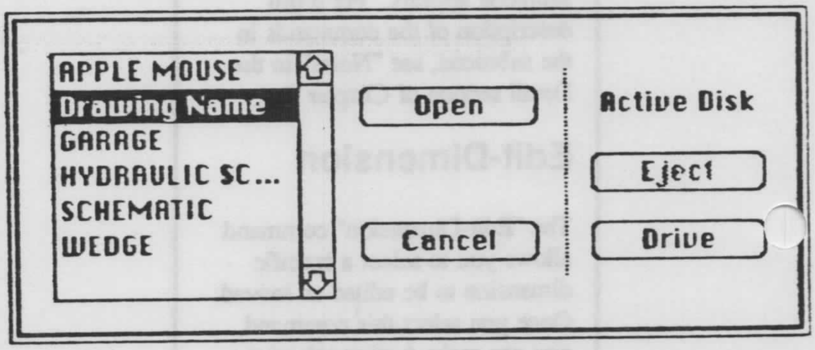
All the commands in this menu, except the last 6, (see chapter 2), are for saving and/or retrieving drawing files or symbol libraries that you have made, or outputting your drawings to a plotter or to EZ-CAM.

New

This command allows you to open a new drawing file after finishing work in another file, without having to reboot the system. After choosing this command and choosing "Enter" the initial screen will appear requesting drawing size. If you complete this command before saving the drawing any unsaved files will be lost.

Open

This command allows you to open previously saved files. When you choose this command a the window shown below will appear. A list of files available on a specific disk will be shown on the left hand side of the window. The name of the active disk appears in the upper right hand corner of the window. To open any of these files select it with the pointer, it will be outlined in black. Then choose "Open" from the list window a that file will be opened. You may select another disk to read files from by choosing "Drive" from the window. If your files are not on a disk currently in the system choose "Eject" and insert the file disk.



Close

Choosing "Close" will close the file you are presently working on, if the file has not been saved then you will be prompted to do so. After closing a file you are free to open a new file, open an exsisting file or "Quit".

Save

This command allows you to file or store a drawing on disk. Choosing "Save" will automatically save the drawing under the curent file name if the file does not have a name the command will default to the "Save as" command.

Save As

This command allows you to name a file before saving it. When you choose "Save As" a window will appear, type in the name for the Drawing.

The disk that will be saved to, will be shown as the "Active Disk". To select another disk, to save to, choose "Drive" from the window. If your file disk is not currently in the system choose "Eject" and insert the file disk.

Choose "Save" and the drawing will be saved under that name on the selected disk and will return you to the drawing when the save is completed. It is suggested that you do "Saves" periodically throughout a work session so, in case of power outage or fatal error, that as little work is lost as possible.

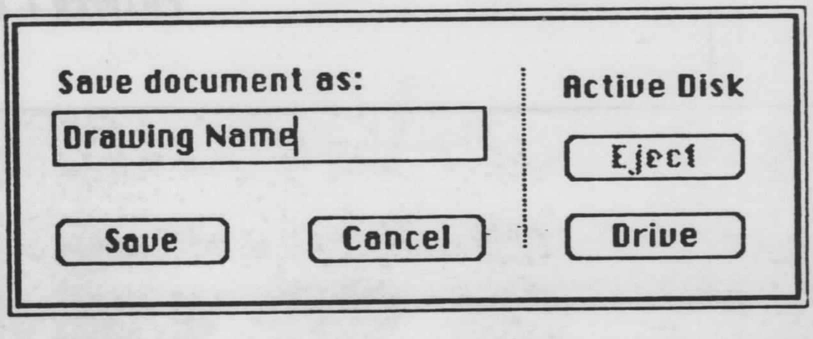
If you choose "Cancel" anytime before saving you will be returned to the drawing. If you choose "Save As" for an exsisting file and rename it, this will file the drawing under the new name and the old drawing will still be filed under the old file name.

Revert To Save

This command enables you to return your drawing to the last saved version. When you choose "Revert To Save" and then "Enter" all the work done since the last save will be lost. You are given the option to cancel using "Return".

Quit

This command is used after you have done a save and want return to the Macintosh Desktop. Choosing "Quit" will close the file you are presently working on, if the file has not been saved then you will be prompted to do so. You are given the option to cancel using "Return".



Save document as: Drawing Name

Read IGES

This command allows you to read an IGES (Initial Graphics Exchange Specification) IGES file into EZ-Draft.

Write IGES

This command allows you to write an IGES (Initial Graphics Exchange Specification) file. The IGES file can then be read by EZ-Cam, or any CAD,CAM or CIM workstation that has an IGES capability.

Plot

This command allows you to create plot files, which are then stored on a disk , and then plot drawings on one of many different plotters.

To actually plot a drawing, you must first call the plot file, using the "Plot" menu from the "File" menu. When the drawing is plotted, it will show everything that was visible onscreen when you last worked with the drawing. Therefore, if you do not want view borders plotted, you must turn them Off with the Border On-Off command.

To create a "Plot" file:

1. Choose "Plot".

2. When the prompt, "Enter name for plot file" appears, enter a name for the plot file. No more than six characters may be entered. When you have finished entering the name of the plot file, choose either "Return" or "Enter", to indicate that you are finished.

The plot file(s) will then be stored on a disk. To plot the files you have saved, see the "Plotting" appendix utility.

Symbol Libraries

Current Symbols

This command allows you to list the names of the symbols in the currently active part. If all of the symbol names do not fit into the prompt area of the screen, choose "Return" to continue your list. The symbols can either be created with the "Create Symbol" command, or placed using the "Symbol Instance" command.

Purge Symbols

This command enables you to "clean out" unused symbols from the database. A prompt will ask if you want to remove any symbols that can be listed with the list command but have no Symbol Instances (i.e. unused in the drawing).

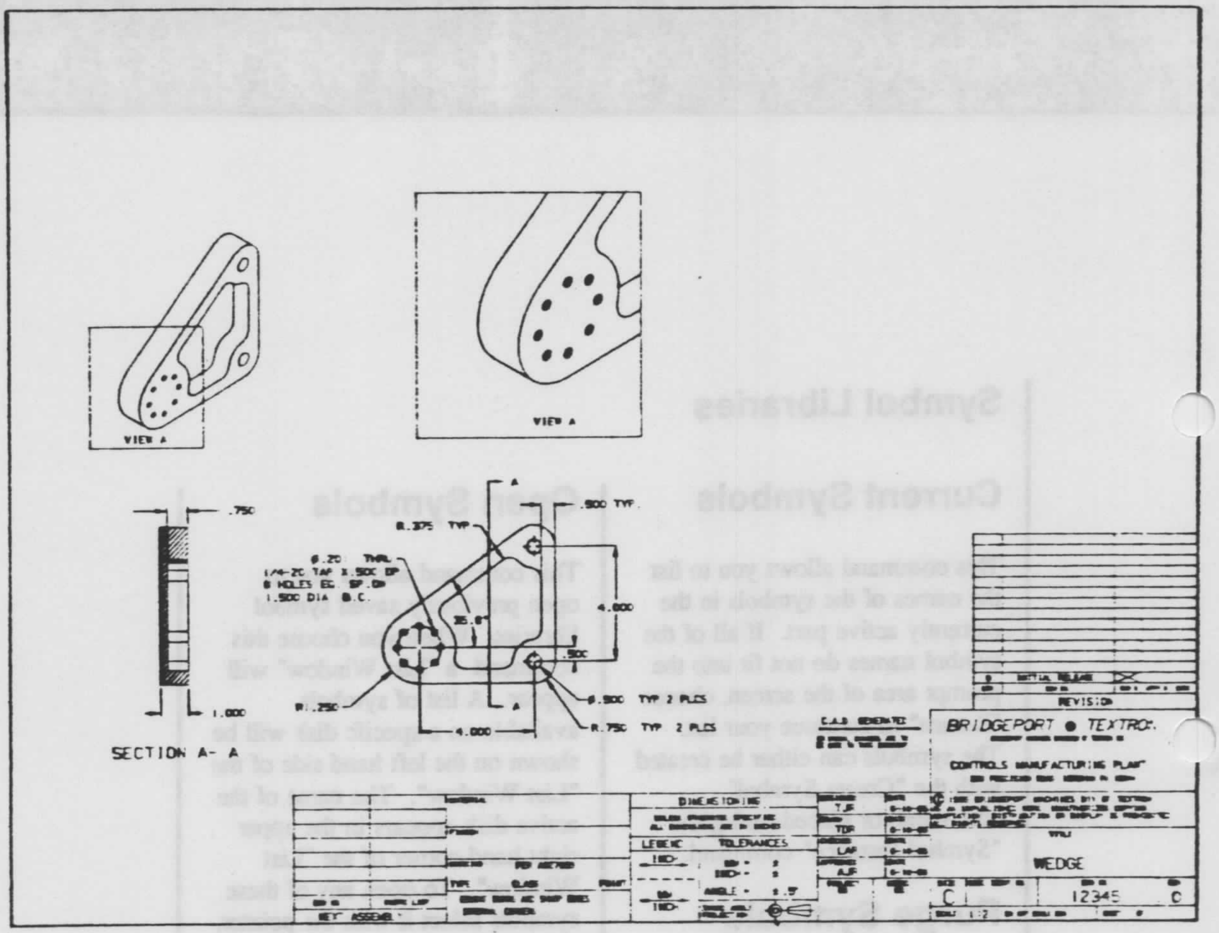
Save Symbol

This command is used to save symbols to a global symbol library that you can store on a separate diskette.

Open Symbols

This command allows you to open previously saved symbol libraries. When you choose this command a "List Window" will appear. A list of symbols available on a specifc disk will be shown on the left hand side of the "List Window". The name of the active disk appears in the upper right hand corner of the "List Window". To open any of these symbols select it with the pointer, it will be outlined in black. Then choose "Open" from the list window and that symbol will be opened. You may select another disk to read symbols from by choosing "Drive" from the "List Window". If your symbols are not on a disk currently in the system choose "Eject" and insert the symbol library disk.

Appendix



EZ-Draft Extras

File Menu

Quit, *Command Q*

Global Menu

Enter, *Double Click Mouse*
Redraw, *Command R*
Zoom All, *Command A*
Erase Last, *Command E*

Explicit Point Entry

X coordinate, *X key*
Y coordinate, *Y key*

Implicit Point Entry

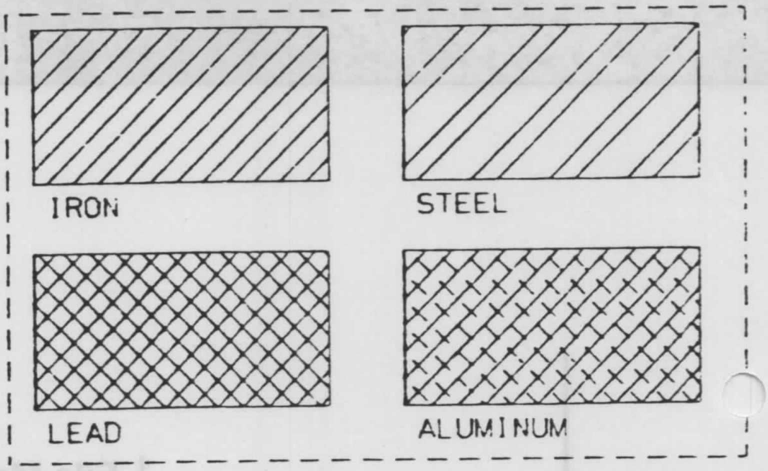
IX (Incremental X coordinate), *Option X*
IY (Incremental Y coordinate), *Option Y*

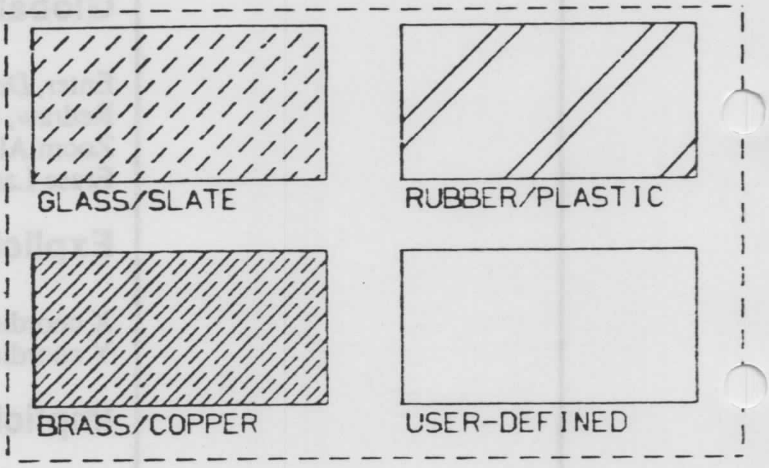
Prompts (yes or no answer required)

"Yes", *"Enter" or Double Click Mouse*
"No", *"Return"*

Shown to the right are the available cross hatching patternson the EZ-Draft system.

1. Iron
2. Steel
3. Lead
4. Aluminum
5. Glass/Slate
6. Rubber/Plastic
7. Brass/Copper
8. User-Defined





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